High Resolution Hybrid RANS/LES Simulations of Flow Over Surface-mounted Cubes Using a Sixth Order Finite Difference Solver

A. Curley\textsuperscript{1}, M. Uddin\textsuperscript{1} and B. Peters\textsuperscript{1}

\textsuperscript{1}North Carolina Motorsports and Automotive Research Center
The University of North Carolina at Charlotte, NC 28223, USA

Abstract
The veracity of two popular turbulence models in predicting highly separated turbulent flows was evaluated in a study of a single surface-mounted cube and two surface-mounted cubes in tandem for various spacings at a Reynolds number of 22,000. Transient hybrid RANS/LES delayed multi-scale simulations using the \( k - \omega \) SST model and the Langtry-Menter SST transition model were performed using a sixth-order finite difference compressible code OVERFLOW on overset chimera grids generated using Chimera Grid Tools. While both turbulence models predicted, within an acceptable engineering accuracy, the mean drag coefficient of an isolated cube and that of the upstream cube in case of tandem configurations, neither model captured the dynamics of the wake accurately. The Langtry-Menter model outperformed the \( k - \omega \) SST model in stagnation regions and provided a closer approximation of the wake evolution despite both models' overall failure in separated regions. However, this improved prediction comes at cost of 20\% increase in computation time.

Introduction
Turbulent flows around single and tandem bluff bodies exhibit complex flow structures of particular interest in engineering applications. Massive separation, multyscale vortical structures, highly turbulent recirculation zones, and periodic vortex shedding in the shear layer result in oscillatory pressure distributions on the surrounding geometry and unsteady flow patterns downstream of the obstacles. Our interest in this area stems from an observation during NASCAR's 2011 Daytona 500 racing event when a new style of racing emerged as two cars locked together bumper-to-bumper exhibited a 20 MPH increase in speed compared to cars spaced out in the standard drafting formation. For those not living in North America, NASCAR is America's prime motorsports sanctioning body, and the Daytona 500 is its most prestigious event. Preliminary computational fluid dynamics (CFD) simulations based on two-equation RANS models at UNC Charlotte accurately predicted this speed increase by an accurate prediction of the reduction in mean drag coefficients; however, these simulations failed to produce a reasonable value for the downforce. Thus, this paper investigates the abilities of the two-equation models to predict flows around a single cubic bluff body and two cubic bluff bodies in tandem arrangement with inter-cube spacing, \( S \), ranging from 0.25\( H \) to 8\( H \).

Surprisingly, literature presents few credible in-depth experimental studies of flow around single bluff bodies. Castro and Robins [1] studied the nature of the wake for a single cube in uniform and turbulent streams with a thick boundary layer and found that the wake decayed within approximately six cube heights downstream. Further investigations by Lyn et al. [3] in the base region concluded that the periodic component was driven by the streamwise size of the vortex formation region while the turbulent component was driven by the wake width. This was based on the observation that the periodic stresses decayed much more rapidly than the turbulent stresses in this region. Observations by Martinuzzi and Tropea [6] of a prismatic obstacle in a fully developed channel flow indicated that for objects with a small aspect ratio, such as a cube, the flow is dominated by the interaction of the horseshoe vortex with the corner vortex behind the cube and with the mixing layer in the wake.

More recent studies have looked at the interaction of two closely spaced bluff bodies. Martinuzzi and Havel [4] found that for small separations (\( S/H \leq 1.4 \)), the shear layer reattaches on the top side of the downstream obstacle. This reattachment results in a separation zone on the front face of the downstream obstacle and a strong stream of fluid directed downward along the front face. As the spacing between the obstacles increases, the flow begins to reattach to the base wall and a second horseshoe vortex in front of the downstream cube emerges. Martinuzzi and Havel [5] later discovered that for spacings between 1.5\( H \) and 2.5\( H \), a geometrically locked regime exists in which the Strouhal number based on the cube spacing remains constant. In this locked regime, the periodic vortex shedding results from the interaction between the side shear layers and the strong vertical stream along the front face of the downstream obstacle.

Computational simulations of \( S > 2H \) have begun to assess the capabilities of turbulence models to predict the characteristic structures in the flow field. Paik et al. [9] reported the failure of unsteady Reynolds-averaged Navier-Stokes (URANS) to capture the horseshoe vortex along the base of the upstream cube and the premature switching to unresolved DNS in refined regions of both detached eddy simulation (DES) and delayed-detached eddy simulation (DDES). For inter-cube spacings \( S < 2H \), a lack of detailed assessment exists for turbulence models and few experimental investigations have reported extensive discussions of the flow field within the cavity region for small spacings.

Numerical Model
The compressible unsteady Reynolds-averaged Navier-Stokes (RANS) equations were solved using a sixth-order finite difference code OVERFLOW (version 2.2g) on overset chimera grids generated using Chimera Grid Tools. The turbulence models implemented in the hybrid RANS/LES multi-scale simulations were the Menter [7] \( k - \omega \) SST model (SST, hereinafter for simplicity) and the Langtry-Menter [2] local correlation-based transition model (LMT). The dimensions of the computational domain replicated the experimental setup of [5] and a detail of the mesh in the immediate region surrounding the cubes is shown in figure 1. The same domain was used for both the single and tandem arrangements.

For the single cube simulations, the mesh consisted of 55 million vertices with an average \( y^+ \) value of 0.3178; for the tandem cube simulations, the mesh consisted of 71 million vertices with an average \( y^+ \) value of 0.3162. For both simulation setups, the immediate area surrounding the cubes (\(-H \leq x \leq 5H \)) was resolved to 0.0125\( H \) between neighboring vertices and a first node height of 0.000277\( H \) for the entire computational domain. The upstream boundary was defined as a uniform velocity inlet and was moved further upstream to \( x = -8H \) to avoid nu-
The introduction of the intermittency factor $\gamma_{sep}$ initiates the production of turbulent kinetic energy ($\tilde{P}_k$) downstream of the transition onset location.

**Results and Discussion**

To evaluate the performance of SST and LMT models in highly separated aerodynamic flows, the flow around a single cubic obstacle was investigated in addition to two cubic obstacles in tandem arrangement. Various flow properties are compared with the experimental studies of [1] and [6]; however, the detailed analysis is limited to the single cube flow in this paper.

CFD predicted drag coefficients, for $0.25 \leq x/H \leq 8$, obtained using SST and LMT models are compared against the experimental results of [5] in Figures 2(a) and (b) respectively. The differences between the predictions from the SST and the LMT models are not very distinguishable. Clearly, the CFD results indicate an inability of both of these models to adequately capture the effects of the upstream cube wake on the downstream cube for inter-cube spacings less than $2H$ and greater than $3H$. For the smaller inter-cube spacings, both models greatly overpredict $C_D$ for the downstream cube; however, as the inter-cube spacing increases, both models begin to underpredict $C_D$ for the downstream cube. For nearly all spacings, both models slightly overpredict the $C_D$ of the upstream cube, but the mean values are within acceptable engineering tolerance; the error is within 5% which is again within the experimental uncertainty.
appears that this profile shifted vertically from the one obtained from the LMT model. Despite the success of both models to emulate \( C_D \), the SST model predicts significantly greater pressure magnitudes on both faces. At this stage it is not clear to us despite this large discrepancies in rear face pressure distribution, why the mean drag predictions from both of these models are very close to the experiment and, as well as, to each other. However, referencing back to our previous studies on NASCAR bump drafting, this inability of the turbulence models to capture the wake correctly contributes to the inaccurate prediction of downforce, but, like this case, predicted the drag values accurately.

<table>
<thead>
<tr>
<th>Experiment [8]</th>
<th>Transition Model</th>
<th>( k-\omega ) SST</th>
</tr>
</thead>
<tbody>
<tr>
<td>( C_D )</td>
<td>1.05</td>
<td>1.026</td>
</tr>
<tr>
<td>Error</td>
<td>—</td>
<td>2.3%</td>
</tr>
</tbody>
</table>

Table 1: Average Single Cube Drag Coefficients

Figure 3: \( C_p \) distribution for a single cube along \( y/H = 0 \) (a) up the front face and (b) up the rear face compared to Castro and Robins [1].

The \( C_p \) distribution along the floor is shown in figure 4 which also contains the experimental pressure distributions from [6]. For this plot, the \( C_p \) values are calculated following the conventions of [6] where the bulk velocity and the freestream static pressure are used as the reference velocity and pressure respectively. Compared to this experiment, the LMT model predicts a reattachment location further downstream. As discussed earlier, the LMT model captures the base pressure at the rear face of the cube, but in the near wake region it begins to underpredict \( C_p \) before asymptoting to the experimental results further downstream. Conversely, the SST model overpredicts the base pressure but then captures the near wake region before diverging from experimental results further downstream.

The mean \( x \)-velocity profiles at three locations along the \( y = 0 \) centerline are shown in figures 5(a)–(c). Both SST and LMT predictions are in close agreement with experimental results; however, very close to the wall as \( z \) approaches zero, the computational results and the experimental results begin to deviate. The good correlation between experimental and computational results of the velocity profiles affirm the effectiveness of a sixth-order solver.

Figure 4: \( C_p \) distribution for a single cube along the floor at \( y/H = 0 \) compared to Martinuzzi and Tropea [6].

Mean turbulent intensity profiles at the same locations as the mean velocity profiles are presented in figures 6(a)–(c). At \( z/H = 0.5 \), both the SST and the LMT models show a much greater turbulent intensity compared to experimental results. The location, however, of the peak turbulent intensity at this position corresponds to that of the experimental results. Additionally, the peak intensity of the LMT model is significantly greater than that of the SST model which illustrates the greater strength of the wake predicted by the LMT model as discussed earlier.

As the streamwise position increases, both turbulence models begin to underpredict the turbulent intensity with decreasing discrepancy between the two models which leads to two observations. First, neither model adequately captures the dissipation of turbulent kinetic energy in the wake region and secondly, the dissipation rate of turbulent kinetic energy for the LMT model decreases more rapidly than that of the SST model. The experimental results of [1] show that the peak turbulent intensity decreases only slightly between \( x/H = 0.5H \) and \( x/H = 2.5H \). Conversely, the peak turbulent intensity decreases by a factor of approximately 1.8 for the SST model and approximately 2.2 for the LMT model over this interval.

The excessive decay of turbulent kinetic energy in the wake is a direct result of the production terms shown in equations (1) and (2) for the SST and LMT models respectively. For the SST model, a careful modification to the \( B^ \) model coefficient may reduce the turbulent kinetic energy dissipation in the wake region. Similarly, the intermittency factor \( \gamma_1 \) controls the production of the LMT model and the model coefficients in equations (3) and (4) may be adjusted to develop a more accurate correlation for highly separated flows. The value of 3.235 in equation (3) was adopted by Langtry and Menter [2] to change the relationship between the vorticity or strain-rate Reynolds number, \( Re_\theta \), and the critical Reynolds number at which intermittency begins to increase, \( Re_{\theta c} \), based on the shape factor at a separation point rather than at that of a Blasius boundary layer. This value was increased from 2.193 for a Blasius boundary layer to the current value of 3.235, effectively reducing the transition onset Reynolds number. Along this thought and the data presented herein, this model coefficient may need to be increased to accurately predict highly separated aerodynamic flows.

Conclusion

The performance of two turbulence models was evaluated in predicting highly separated aerodynamic flows. It was found that the Langtry-Menter transition model reproduced \( C_p \) distribution along the front face of single cube and overpredicted \( C_p \).
in the large separation region along the rear face while the $k-\omega$ SST model significantly overpredicted $C_p$ for both faces. This effect was attributed to an underprediction of turbulent intensity caused by an overprediction of turbulent kinetic energy dissipation in the wake region behind the upstream cube. The data presented herein offers an explanation of the dilemma in previous CFD simulations that reproduced the mean drag coefficient of two vehicles in tandem drafting but failed to reproduce reasonable downforce predictions. The Langtry-Menter transition model presents a baseline for a new physical transition onset modeling; however, adjustments to the model correlations are required to suit the flow structure development in highly separated flows.

Acknowledgment
Alex Curley acknowledges the support from UNC Charlotte Levine Scholars Program. Support from UNC Charlotte University Research Computing and Mosaic Computing are also gratefully acknowledged.

References