RANS Simulations of a Simplified Tractor/Trailer Geometry

Christopher Roy, Jeffrey Payne, Mary McWherter-Payne, and Kambiz Salari

1MS 0825, PO Box 5800, Sandia National Laboratories
2L-228, PO Box 808, Lawrence Livermore National Laboratory

Abstract

Steady-state Reynolds-Averaged Navier-Stokes (RANS) simulations are presented for the three-dimensional flow over a simplified tractor-trailer geometry at zero degrees yaw angle. The simulations are conducted using the SACCARA multi-block, structured CFD code. Two turbulence closure models are employed: the one-equation Spalart-Allmaras model and the two-equation k-ω model of Menter. The discretization error is estimated by employing two grid levels: a fine mesh of approximately 20 million grid points and a coarse mesh of approximately 2.5 million grid points. Simulation results are compared to the experimental data obtained at the NASA-Ames 7x10 ft wind tunnel. Quantities compared include: surface pressures on the tractor/trailer, vehicle drag, and time-averaged velocities in the base region behind the trailer. The results indicate that both turbulence models are able to accurately capture the surface pressure on the vehicle, with the exception of the base region. The Menter k-ω model does a reasonable job of matching the experimental data for base pressure and velocities in the near wake, and thus gives an accurate prediction of the drag. The Spalart-Allmaras model significantly underpredicted the base pressure, thereby overpredicting the vehicle drag.

Introduction

In a typical class 8 tractor/trailer, power required to overcome rolling resistance and accessories increase linearly with vehicle speed, while energy losses due to aerodynamic drag increase with the cube of the speed. At a typical highway speed of 70 mph, aerodynamic drag accounts for approximately 65% of the energy output of the engine (McCallen et al 1999). Due to the large number of tractor/trailers on the US highways, even modest reductions in aerodynamic drag can significantly reduce domestic fuel consumption. Lower fuel con-
sumption will result in a reduction in pollution emissions, and, more importantly, a reduced dependence on foreign oil.

The most common turbulence modeling approach for engineering applications involves solving the Reynolds-Averaged Navier-Stokes (RANS) equations. With this approach, the effects of the inherently three-dimensional and time-varying turbulent eddies on the mean flow are modeled and not simulated. These effects of the turbulence, namely increased transport of momentum and energy, are incorporated via the eddy viscosity and eddy conductivity, respectively. In general, it is desirable to obtain steady-state solutions to the RANS equations; the simulation of unsteady RANS flows may only be valid when there is a clear separation between the unsteady scales and the turbulent scales.

RANS turbulence models were generally developed to solve simple, zero pressure gradient attached flows. These models often fail in the presence of large pressure gradients and/or separated flow regions. While the flow over the major part of a tractor/trailer is attached and therefore amenable to RANS modeling, the flow in the base region involves separation off of the rear end of the trailer. This recirculation zone is generally unsteady, with large-scale turbulent structures shedding from the edges. Accurate prediction of the flow in the base region is important since it determines the pressure on the trailer base. The pressure drag is the primary component of the overall aerodynamic drag for tractor/trailer configurations, and small errors in the predicted base pressures can significantly affect the drag calculations. The goal of this study is to assess the ability of steady-state RANS turbulence models to accurately predict the flowfield and aerodynamic drag for tractor/trailer configurations.

Problem Formulation

The configuration to be examined is the Ground Transportation System (GTS) studied experimentally at the NASA Ames research center (Storms et al 2001). The GTS geometry is a simplified tractor/trailer configuration which is mounted on four posts in the wind tunnel. A photograph of the GTS in the NASA Ames 7x10 ft wind tunnel is shown in Figure 1. The GTS model is an approximately 1/8 scale class 8 tractor/trailer configuration. The Reynolds number based on the trailer width (W=0.3238 m) is 2 million, approximately one-half of full scale.
The GTS geometry, including the wind tunnel walls, is discretized using two mesh levels: a coarse mesh using 2.5 million grid points and a fine mesh using 20 million grid points. The grids are generated such that the wall $y'$ values on the truck surface, supports, and lower wind tunnel wall are everywhere less than unity on the fine mesh. The side and top wind tunnel walls employ slip flow conditions. Structured meshes are employed using point-to-point match up at the block boundaries. The coarse mesh is domain-decomposed into 125 zones and is shown in Figure 2. The fine mesh is decomposed into 1149 zones. Both the coarse and fine meshes are run on the massively parallel ASCI Red machine using one processor for each zone. The axes employed in the current effort are shown Figure 2, with the x axis starting at the front of the tractor and running downstream, the y axis in the vertical direction, and the z axis starting at the GTS symmetry plane and running spanwise towards the side wall.

In order to ensure that the simulated flow matches closely with the flow in the wind tunnel, a number of freestream conditions are matched. First, the inflow plane is set with the appropriate stagnation conditions of the tunnel shown in Table 1. The back pressure at the simulated outflow plane is then
adjusted until the reference pressure located at \((x/W=4.47, y/W=2.59,\) and \(z/W=-4.7)\) on the tunnel side wall reaches the wall reference pressure given in the table. The pressure coefficient on the wind tunnel side-wall (coarse grid simulation) is compared to experimental values in Figure 3. The boundary layer on the bottom wall was measured in a tunnel-empty configuration. Sample boundary layer profiles upstream of the GTS model from the simulation using the Menter k-\(\omega\) model are compared to the tunnel empty-profile from the experiment in Figure 4. The simulation predicts a profile similar to the experiment, at least until the presence of the GTS model is seen at approximately 0.7 m upstream. The GTS surface, the posts, and the wind tunnel floor are modeled with adiabatic, no-slip boundary conditions, while the tunnel top and side walls employ a slip boundary condition.

Table 1. Freestream conditions used in the simulations

<table>
<thead>
<tr>
<th>Tunnel Condition</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Stagnation Pressure</td>
<td>102,653.2 N/m²</td>
</tr>
<tr>
<td>Stagnation Temperature</td>
<td>282.06 K</td>
</tr>
<tr>
<td>Wall Reference Pressure</td>
<td>97,582.2 N/m²</td>
</tr>
<tr>
<td>Reference Mach number</td>
<td>0.27</td>
</tr>
<tr>
<td>Back Pressure</td>
<td>100,136.0 N/m²</td>
</tr>
<tr>
<td>Tunnel Floor BL Thickness</td>
<td>0.053 m</td>
</tr>
<tr>
<td>Wall Temperature BC</td>
<td>adiabatic</td>
</tr>
</tbody>
</table>

Figure 3. Wind tunnel side-wall pressure
RANS Simulations of a Simplified Tractor/Trailer Geometry

Numerical Formulation

CFD Code

The CFD code is SACCARA, the Sandia Advanced Code for Compressible Aerothermodynamics Research and Analysis, and was developed from a parallel distributed memory version (Wong et al. 1995) of the INCA code, originally written by Amtec Engineering. This code is used to solve the Navier-Stokes equations for conservation of mass, momentum, energy, and turbulence transport in three-dimensional form. The governing equations are discretized using a cell-centered finite-volume approach. A finite-volume form of Yee’s symmetric TVD scheme (Yee 1987) is employed. This flux scheme is second-order accurate and reduces to a first-order Roe-type flux (Roe 1981) in regions of large gradients based on a minmod limiter. The viscous terms are discretized using central differences. The SACCARA code employs a massively parallel distributed memory architecture based on multi-block structured grids. The solver is a Lower-Upper Symmetric Gauss-Seidel scheme (Yoon and Jameson 1988) which provides for excellent scalability up to thousands of processors. A number of code verification studies have been performed which give confidence that the code is free from coding mistakes including comparison to established numerical benchmark solutions and code to code comparisons (Roy et al 2000). The fine grid and coarse grid simulations presented herein were run in parallel on the ASCI Red parallel processing machine using 1149 and 125 processors, respectively.
Turbulence Models

Two turbulence models are examined in the current work: the one-equation eddy viscosity transport model of Spalart-Allmaras (Spalart and Allmaras 1994) and the Menter k-ω model (Menter 1994). The Spalart-Allmaras model requires the solution of a single transport equation for the eddy viscosity. The Spalart-Allmaras model has proven to be a numerically robust approach, and generally good results have been demonstrated for a wide variety of flows. The Menter k-ω model is a hybrid model which uses a blending function to combine the best aspects of both the k-ω and the k-ε turbulence models. Near solid walls, a k-ω formulation is used which allows integration to the wall without any special damping or wall functions. Near the outer edge of the boundary layer and in shear layers, the model blends into a transformed version of the k-ε formulation, thus providing good predictions for free shear flows. This model also shows less sensitivity to freestream turbulence quantities than other k-ω formulations. In both cases, the turbulence models were integrated to the wall in order to avoid model validation issues associated with wall functions.

Numerical Accuracy

Before the simulation results can be compared to experimental data, the numerical accuracy of the solutions must be assessed. For the steady-state Menter k-ω computations examined herein, the solution accuracy will be judged by examining the iterative convergence of the solutions to steady-state as well as the discretization error. The numerical accuracy of the Spalart-Allmaras results is expected to be comparable.

Iterative Convergence

The solutions are marched (iterated) in pseudo-time until a steady-state answer is obtained. Iterative convergence is assessed by examining the steady-state residuals of the momentum equations. The steady-state residual is defined by plugging the solution at the current iteration into the discretized form of the steady-state governing equations (omitting the time derivative). The iterative error in the solution tends to drop in a similar fashion as the residual. The coarse grid solutions were converged by approximately seven orders of magnitude, while the fine grid solutions were converged by approximately five orders of magnitude. The limited convergence of the fine grid solutions may introduce some small iterative error in the results, and will be converged further in the final paper.
Discretization Error

The discretization error is estimated by generating solutions on two mesh levels. Since the coarse mesh is determined by eliminating every other gridline from the fine mesh, the grid is consistently refined throughout the entire domain, and Richardson extrapolation can be used to estimate the exact solution. This extrapolated solution is then used to judge the error in the fine grid solutions. The coarse and fine grid results for surface pressure on the front of the tractor are presented in Figure 5 for the Menter k-ω model. While the results do show some sensitivity to grid refinement, the estimated error in the fine grid solution is approximately ±0.05 ΔCp. Coarse and fine grid results for the base of the trailer are shown in Figure 6, with the maximum estimated error to be less than ±0.01 ΔCp (note the expanded scale for ΔCp). The Spalart-Allmaras model was only run on the fine mesh; however, the spatial discretization error is expected to be similar to that of the Menter model.

Figure 5. Pressure on front of tractor (coarse and fine meshes)

Figure 6. Pressure on base of trailer (coarse and fine meshes)
Surface Comparisons

Surface Pressure

The surface pressure for the two turbulence models using the fine mesh is compared to experimental data (Storms et al 2001) on the front of the tractor in Figure 7. Both simulations show good agreement with the experimental data. Simulation results are presented for the base of the trailer in Figure 8. In this case, the Menter k-ω model does a reasonable job of matching the pressure levels, while the Spalart-Allmaras model significantly underpredicts the pressure on the base.

Aerodynamic Drag

Aerodynamic drag predictions using the two turbulence models as well as the experimentally measured drag are presented in Table 2. These drag results
(both predicted and experimental) are for the GTS model only and do not include the support posts. The Menter k-ω results are approximately 7.5% higher than the experimental value, while the Spalart-Allmaras results are nearly 50% high. The overprediction of the drag with the Spalart-Allmaras model is due to the poor prediction of the base pressure. Also shown in the table is the estimated uncertainty in the experimental drag coefficient (Storms et al 2001), and the estimated numerical error for the fine grid Menter k-ω simulation. This numerical error estimate is determined by performing Richardson extrapolation using the coarse and fine grid drag coefficients, 0.474 and 0.298, respectively. The resulting extrapolated value for the drag coefficient using the Menter k-ω model is 0.239, which is essentially within the experimental uncertainty bounds.

Table 2. Drag coefficients

<table>
<thead>
<tr>
<th></th>
<th>Drag Coeff., $C_D$</th>
<th>Estimated Uncertainty/Error</th>
</tr>
</thead>
<tbody>
<tr>
<td>Experiment (Storms et al 2001)</td>
<td>0.25</td>
<td>±0.01</td>
</tr>
<tr>
<td>Menter k-ω</td>
<td>0.298</td>
<td>±0.06</td>
</tr>
<tr>
<td>Spalart-Allmaras</td>
<td>0.413</td>
<td>--</td>
</tr>
</tbody>
</table>

Field Comparisons

Vertical Streamwise Cut

Velocity data are available from PIV measurements performed at the NASA Ames 7x10 ft wind tunnel (Storms et al 2001). These PIV data represent a time-averaged picture of the flow in the wake regions immediately behind the trailer base. Figure 9 gives streamlines based on the PIV data in a vertical streamwise cut through the wake ($z/W = 0$). The flow is from left to right, with the base of the trailer shown on the left; the PIV window is also shown in the figure. A large, counter-clockwise-rotating vortex is centered at approximately $x/W = 8$, $y/W = 0.4$. Also, the presence of a clockwise-rotating vortex is suggested by the vertical nature of the streamlines in the upper right-hand corner of the PIV window. A similar view of the streamlines from the fine grid computations using the Menter k-ω model is shown in Figure 10. These Menter k-ω computations predict a more symmetric pair of vortices than is indicated from the experimental data. The location of the experimental PIV window is shown in the figure for reference. Streamlines for the Spalart-Allmaras model are presented in Figure 11, showing a much shorter recirculation zone than the Menter model. This shorter recirculation zone produces larger velocities in the outer flow as the flow accelerates around the wake. The higher velocities result in lower pressures and hence higher drag (see Table 2).
Figure 9. Experimental streamlines: vertical streamwise cut ($z/W = 0$)

Figure 10. Computational streamlines: vertical streamwise cut ($z/W = 0$)

Figure 11. Computational streamlines: vertical streamwise cut ($z/W = 0$)
Horizontal Streamwise Cut

Experimental PIV results for a horizontal streamwise cut through the wake (y/W = 0.696) is shown in Figure 12. Two counter-rotating vortices are evident at x/W = 8.5. The computational streamlines for this case using the Menter k-ω model are given in Figure 13. In this case, the location of the vortices is accurately predicted by the RANS model.

Conclusions

Steady-state RANS simulations were conducted for the flow over the GTS geometry. The numerical accuracy of the computed flowfields was assessed by performing the computations on multiple grids. Simulation results using the Menter k-ω turbulence model gave good agreement with the experimental data for surface pressure, field velocities in the wake, and drag coefficient; however, this model predicted a more symmetric pair of counter-rotating vor-
tices in the vertical streamwise plane than was indicated in the experiment. These results suggest that the Menter k-ω model can accurately predict the drag for tractor/trailer configurations when performed on highly refined meshes using integration to the wall, although some time-averaged flow structures are not captured. Simulation results with the Spalart-Allmaras turbulence model showed good agreement with experimental data for the surface pressure in the attached flow regions, but significantly underpredicted the pressure in the base region. As a result, the Spalart-Allmaras model overpredicted the drag coefficient by nearly 50%.

References


Spalart PR, Allmaras SR. (1994) A one-equation turbulence model for aerodynamic flows. La Recherche Aerospatiale 1: 5-21


