Aerodynamics Overview of the Ground Transportation Systems (GTS) Project for Heavy Vehicle Drag Reduction*

Walter T. Gutierrez, Basil Hassan, Robert H. Croll and Walter H. Rutledge

Sandia National Laboratories, Albuquerque, NM 87185

RECEIVED

DEC 27 1995

OSTI

ABSTRACT

The focus of the research was to investigate the fundamental aerodynamics of the base flow of a tractor trailer that would prove useful in fluid flow management. Initially, industry design needs and constraints were defined. This was followed by an evaluation of state-of-the-art Navier-Stokes based computational fluid dynamics tools. Analytical methods were then used in combination with computational tools in a design process. Several geometries were tested at 1:8 scale in a low speed wind tunnel. In addition to the baseline geometry, base add-on devices of the class of ogival boattails and slants were analyzed.

NOMENCLATURE

a	Speed of sound
A	Body axis axial force (positive parallel to the positive x_{axis} direction, Figure 5)
C_A	Axial force coefficient, $C_A = A/q_{\infty}S$
C_D	Wind averaged drag coefficient, Reference 17
C_l	Rolling moment coefficient, $C_l = l/q_{\infty}Sw$
C_m	Pitching moment coefficient, $C_m = m/q_{\infty}Sw$
C_n	Yawing moment coefficient, $C_n = n/q_{\infty}Sw$
C_N	Normal force coefficient, $C_N = N/q_{\infty}S$
C_P	Static pressure coefficient, $C_P = (P - P_{\infty})/q_{\infty}$
C_Y	Side force coefficient, $C_Y = Y/q_{\infty}S$
h	Trailer height (measured to bottom of tires), 4.11 m (13.50 ft) full scale
l	Body axis rolling moment (positive moment causes clockwise roll about MRP, driver's view looking forward)
m	Body axis pitching moment (positive moment causes nose up about MRP, driver's view looking forward)
M	Mach number, $M = V/a$
MRP	Body axis moment reference point, center of the rear axles of the tractor at ground level ($x/w = 2.451$, $y/w = -0.196$, $z/w = 0.0$)
n	Body axis yawing moment (positive moment causes nose right about MRP, driver's view looking forward)

N	Body axis normal force (positive parallel to the positive y_{axis} direction, Figure 5)
P	Static pressure
q	Dynamic pressure, $q = 0.5\rho V^2$
Re_w	Reynolds number, $Re_w = \rho V w / \mu$
S	Reference area, $S = hw = 10.66 \text{ m}^2 (114.75 \text{ ft}^2)$ full scale
V	Velocity
V_{yz}/V_{∞}	Velocity projected in y_{axis} - z_{axis} plane, Figure 5, non-dimensionalized by freestream velocity
w	Trailer width, 2.59 m (8.50 ft) full scale
x, y, z	Body axis cartesian coordinates
x'/w	Body axial distance behind trailer (non-dimensionalized) as measured parallel $+x_{axis}$ direction, Figure 5, (e.g., when $x/w = 7.65$, $x'/w = 0$)
Y	Body axis side force (positive parallel to the negative z_{axis} direction, Figure 5)
ρ	Density
μ	Viscosity
Ψ	Yaw angle

Subscripts

∞	freestream value
----------	------------------

INTRODUCTION

Aerodynamic prediction for ground transportation vehicle external design requires a thorough understanding of the complex flow field within the viscous region of the boundary layer, and the swirling vortices in the wake, extending to the far field of the freestream. While all aerodynamic problems are different, most benefit from a combination of analytical, computational, and experimental analysis. Included is an overview of the application of these analysis tools in a methodical manner drawing upon the strengths of each technique. The result is a case study design as well as data useful for future research on ground transportation vehicles.

The focus of the research was increasing the knowledge level of fluid flow management in the base region of Class-8 van-type tractor trailers. Within the last twenty years, considerable effort has been expended on successfully reducing the drag of the front region of the tractor trailer. However, few if

MASTER

* This work was supported by the United States Department of Energy under Contract DE-AC04-94AL85000.

any economically feasible designs have transitioned from research and development to the commercial market to improve the management of the trailer wake flow. A 15% reduction in drag would result in a 5% reduction in fuel costs, which equates to an \$800 fuel savings for the average Class-8 tractor trailer per year.¹ While the 15% value is near optimum and difficult to achieve, a 5 to 10% reduction is very possible.

Initially, industry design needs and constraints were defined. This was followed by an evaluation of state-of-the-art Navier-Stokes based computational fluid dynamics tools. Analytical methods were then used in combination with computational tools in a design process. Several geometries were tested at 1:8 scale in a low speed wind tunnel. The final step was comparison and analysis of results.

The Sandia National Laboratories Ground Transportation System (GTS) Baseline vehicle shape utilized for the research is shown in Figure 1. It is typical of a near future tractor trailer as shown by the insert of the Penske Racing vehicle. Represented within this design is a typical cab-over tractor with a van-type trailer. In addition to the baseline geometry, several base add-on devices were analyzed. The first class of geometries were ogival boattails (Figure 2 and Figure 3) that lend toward the study of boundary layer separation. The second class of geometries were slants (Figure 4), similar in design to the Ahmed, et al.² add-on slants, that lend toward the study of vortex driven flows.

Results are both procedural and quantitative. Many lessons were learned in terms of the strengths and weaknesses of the analytical, computational, and experimental tools. In addition, results and comparisons of the aerodynamic characteristics of the geometries will be presented. Included are performance data, such as axial force coefficients, as well as design data such as surface pressure contours and wake velocity profiles. References 3 and 4 provide further detail discussion and results of the Sandia National Laboratories computational and experimental studies, respectively.

INDUSTRY NEEDS AND DESIGN CONSTRAINTS

Prior to initiation of the design process, restrictions and exclusions were studied pertaining to constraints imposed by federal laws, city driving, and functionality.

PRIME ADD-ON DEVICE DESIGN CONSTRAINT - As stated by the Department of Transportation, the maximum length exclusion for an aerodynamic device for a GTS type vehicle is currently 1.5 m (5 ft^{5,6}).

"Aerodynamic devices (air deflectors) are excluded from the measurement of trailer length. They shall not obscure tail lamps, identification lamps, license plates or any other required safety device such as hazardous materials placards. These devices shall not extend beyond 5 ft from the rear of a trailer when in the operational position."

UNINTENDED CONSEQUENCES OF DEVICES - Although the aerodynamic drag of tractor trailers has been significantly reduced in the past twenty years, it has not been without some compromise to the trucking industry. A study group of The Maintenance Council (TMC) of The American Trucking Associations investigated the impact on maintenance operations due to tractor mounted aerodynamic fuel saving devices⁷.

Problems noted included:

- components operating at higher temperature due to altered intake flow
- debris buildup on wiring and connectors
- corrosion due to retained moisture in enclosed areas
- reduced visibility
- damage at loading docks
- poor mounting hardware
- increased wind buffeting effects
- reduced access to maintenance items
- increased tractor trailer weight

Design must be conducted with the knowledgeable caution of the above mentioned items, in addition to ones specific to trailer base add-on devices. This additional list might include:

- rear underride
- visibility of lights and license plate
- collection of water and road debris
- material dynamic response (e.g., panel flutter) due to road speed and wind excitation
- material wear due to road conditions and ultra-violet light exposure

ASPECTS OF THE COMMERCIAL MARKET - Most aerodynamic devices currently in operation are effective in terms of up-front costs; however, some devices are not cost effective when their life cycle cost is estimated and parameters like additional maintenance, effects on neighboring components, and reduced accessibility are included. The trucking community expects the add-on drag reduction device to pay back the up-front unit cost and installation cost in one year of service, with minimal additional life cycle costs. Important in bringing drag reduction devices successfully to market is a favorable life cycle economic impact for a particular tractor trailer combination used in a particular environment.

EVALUATION OF CFD TOOLS

The goal of the computational fluid dynamics (CFD) evaluation was to establish computational needs for low speed aerodynamics of ground transportation systems. Then, to acquire and evaluate the tools necessary for solid model definition, grid generation, numerical computation, and post-process analysis and presentation. Consideration was given to defining the boundary conditions of symmetry planes, far field planes, and moving ground planes. Finally, benchmark solutions were run to compare with existing data from the literature.

The following criteria were utilized for evaluation:

- Accuracy - the code must be proven with published comparisons to experimental results or other numerical techniques
- Pre/post process - the code must have a proven and efficient process (or compatibility with other codes) for solid model generation, grid generation, and analysis
- Speed - the code must demonstrate relatively fast convergence when compared to other comparable techniques
- User base - it was desired that the code have a multiple user experience base at Sandia
- Portability - the code must be able to run on several classes of computers, including desk top workstations, multi-processing machines, and supercomputers

- Boundary conditions - the code must have a flexible array of boundary conditions to model the problems of interest, including simulating a moving boundary or ground plane

Codes were chosen and subdivided by the type of computational grid used to solve the governing equations. Structured grids have been the most widely used in CFD and can be produced around complex shapes by subdividing the grid into blocks of grid cells. Resolving regions of large flow gradients such as shock waves, surface boundary layers, and free shear layers can be accomplished by clustering the grid cells within the multi-block structure. However, constructing such multi-block grids and relaying information on how each block is connected to others in the flow solver can be time consuming. On the other hand, grid generation around complex shapes using unstructured grids is relatively simple and requires significantly less time than required to construct a multi-block structured grid. However, unstructured grids (tetrahedrons) do have a significant drawback. They do a relatively poor job of resolving regions of high flow gradients when compared to structured grids (particularly boundary layers). Capturing these effects can be improved by including more computational cells in these regions, but then computational costs go up as the number of cells is increased. These two differences, ease of grid generation and gradient resolution, are major concerns in the evaluation process.

Another issue of consideration is the type of flow solver that is used. For the conditions of interest of a tractor trailer traveling on the highway at 29 m/s (60 m.p.h.), the freestream Mach number is on the order of 0.1, and the air flow is essentially incompressible, i.e., the density of the air is constant. Traditionally, standard compressible flow solvers, which allow the density to change, have had trouble computing flows where the Mach number is less than 0.2-0.3. The primary problem is slow convergence times for solution of the Navier-Stokes equations, which are a statement of conservation of mass, momentum and energy. As the Mach number drops, the density becomes a constant value and the time derivative of the density in the continuity equation disappears. The result is that the continuity equation becomes decoupled from the momentum equations, which results in a weak relation between the pressure and density. To circumvent this problem at incompressible Mach numbers, an artificial compressibility condition is imposed. The idea is to instead solve a modified continuity equation for the pressure. The equation is solved such that when the time-like derivative of the pressure is zero, continuity is satisfied. Solving the above modified continuity equation rather than the standard continuity equation is commonly referred to as a pressure-based technique.

The code that was chosen was RAMPANT⁸, a commercial computational fluid dynamics program for predicting fluid flow around complex geometries. For the cases to be presented, RAMPANT's incompressible option was used. RAMPANT uses the artificial compressibility technique to solve the Navier-Stokes equations on unstructured grids; allowing either triangular or quadrilateral elements in two-dimensions and tetrahedral or hexahedral elements in three-dimensions. For all calculations, the Renormalized Group (RNG) k- ϵ turbulence model in RAMPANT was employed. The steady-state solutions were converged to second-order spatial accuracy.

In order for the CFD code to be utilized, it was necessary to

have both pre- and post-processing tools in place. First a surface definition of the geometry of interest had to be constructed. It was decided to use the Pro/ENGINEER⁹ software developed by Parametric Technology Corporation. This code is widely used at Sandia and has the flexibility to define the surface geometry for both computational as well as for experimental applications. The code also allows for alterations of surface dimensions as well as capabilities to add additional parts to the geometry. For instance, an additional piece may be a boattail device attached to a baseline vehicle shape.

With the surface geometry defined, the computational grids must be created. The RAMPANT grid generation software, preBFC¹⁰, was used for two-dimensional grids. The PATRAN¹¹ software package was used to create the three-dimensional triangular surface grids that were input into RAMPANT's TGrid¹² software for tetrahedral volume grid generation.

RAMPANT's own module was used for post-processing the unstructured grid solutions. A specialized plotting analysis code, TECPLOT¹³, has some limited capability for doing unstructured grid post-processing, and proved useful for comparing the CFD results with analytical and experimental prediction techniques.

Benchmark two-dimensional and three-dimensional cases were computed with RAMPANT and compared with experimental data available within the literature, as well as with solutions generated with other CFD tools in use at Sandia. Results compared favorably in terms of performance characteristics, surface pressure distributions, and wake definition.

GTS GEOMETRY DEFINITION

The research vehicle for the studies is shown in Figure 1 and Figure 5 and is designated the Sandia National Laboratories Ground Transportation System (GTS) Baseline vehicle. The shape is representative of a cab-over tractor trailer design, similar to the Penske vehicle also shown in Figure 1. Details such as wheel wells, mirrors, and the gap between the tractor and trailer have been compromised for the sake of simplicity. Since primary concern was with the base of the trailer, and changes in drag that would result from add-on devices, it was felt that a representative streamlined shape with qualitative details would be adequate. The vehicle length is 19.8 m (65 ft) with a base trailer width of 2.6 m (8.5 ft) and a base trailer height (bottom of tires to top of trailer) of 4.1 m (13.5 ft). Dimensions shown in Figure 5 are non-dimensionalized with the base trailer width, w .

CONCEPTUAL DESIGN PROCESS

The design process goal was to utilize existing literature, analytical techniques, computational tools, and experimental methods to increase the state-of-the-art knowledge of flow in the vicinity of the base region of a heavy ground transportation vehicle.

EXISTING LITERATURE - A NASA report¹⁴ concentrated upon defining a near optimum boattail geometry for a high volume type vehicle like that representative of the GTS Baseline. It also addressed truncating the shape near the separation point, in an effort to minimize boattail length. The

NASA results were based upon a combination of full scale coast-down testing and wind tunnel testing.

When scaled to the GTS vehicle, the resulting geometry was an ogive contoured boattail, tangent at the base of the trailer, and truncated at an axial location 1.6 m (5.3 ft) behind the trailer, with a truncated base diameter of 1.6 m (5.1 ft).

SANDRAG - A set of parametric calculations on the drag of various boattails on a body of revolution were conducted using an incompressible drag prediction code SANDRAG^{15,16}. The purpose of this study was to provide some initial analytical guidance regarding the optimal shape of add-on boattail type drag reduction devices within design constraints. SANDRAG was originally designed to predict the pressure distribution and drag of bodies of revolution at zero degree angle-of-attack in incompressible flow. The approach taken is the classical coupling of the potential flow solution with the boundary layer solution. Empirical correlations were developed to estimate the base-pressure coefficient on a wide variety of geometries. The most obvious restriction of the SANDRAG code is the limitation to bodies of revolution at zero degree angle-of-attack in incompressible flow. In addition, it cannot model objects with blunt forebodies with diameter discontinuity, or bodies with steep axial slope; however, it can provide guidance for trends.

The SANDRAG shape representative of the GTS Baseline, denoted 'SANDRAG Vehicle' in Figure 6, resulted in a predicted tail axial force coefficient (integrated pressure over ogive and base) contribution of $C_A = 0.13$ ($\Psi = 0^\circ$). This is consistent with Reference 17 which suggests a wind averaged drag coefficient of $\bar{C}_D(90 \text{ km/hr}) = 0.14$ as being representative of the rear pressure losses of a typical heavy vehicle commercial road vehicle. Even though the SANDRAG value is at 0° yaw, and the reference value is for the weighted range of yaw angles consistent with a wind averaged drag coefficient, this gives a rough indication of the validity of the SANDRAG code for application to this design process.

A parametric study of elliptical, ogival, and conical shapes (some with truncated bases) was then conducted on the SANDRAG Vehicle (e.g., Figure 6). The study was influenced by the current Federal mandate that drag reduction devices not extend beyond 1.5 m (5 ft) from the rear of a trailer when in the operational position. However, on the assumption that the '5 ft' rule might be extended in the future to allow for more efficient devices⁶, the problem was bounded by considering lengths as long as 2.4 m (8 ft).

SANDRAG results indicate the maximum drag reduction for a 1.5 m (5 ft) long cone or ogive is achieved with a 1.8 m (6 ft) diameter base truncation, with the conical boattail producing the lowest drag. A 2.4 m (8 ft) long cone was optimized with a 1.8 m (6 ft) diameter base truncation, while a 2.4 m (8 ft) long ogive was optimized with a 1.5 m (5 ft) diameter base truncation, with the ogive boattail producing the lowest drag. An elliptical boattail would have to be greater than 2.4 m (8 ft) to be useful for the current application.

ADD-ON GEOMETRIES: OGIVES AND SLANTS - Based on the open literature and SANDRAG results, it was decided to design two boattails. One a 1.5 m (5 ft) long ogival boattail with a truncated base diameter of 1.8 m (6 ft) (Figure 2). The second a 2.4 m (8 ft) ogival boattail with a truncated base diameter of 1.5 m (5 ft) (Figure 3). The ogive surfaces are tangent at the top of the trailer and on the sides.

The blend is from a 2.6 m (8.5 ft) square at the base of the trailer to a virtual square at the point of truncation, with corner vertices that have a variable radius that is sharp at the trailer base and transition to form a near circle at the truncation point. Due to limitations in the solid modeling program, the radius was defined perpendicular to the curved corner vertices, thus the resulting base is not a true circle. However, the deviation is negligible aerodynamically, and for modeling purposes can be constructed either way.

For the 5 ft boattail, it was decided to consider the ogive instead of the conical shape for three reasons. First, the noted NASA research states the optimum boattail shape to be ogive contoured, tangent at the base of the vehicle, and truncated near the separation point. Second, most economically manufacturable boattails will be of the inflatable type. Due to the hoop stress while inflated, i.e., pressure induced deformation of the flexible material, a conical shape would be difficult to maintain without stiffeners. Finally, when boundary layer separation occurs near the base of a conical boattail, the separation tends to propagate abruptly all the way back to the leading edge, and can significantly increase the drag. However, on ogival boattails, if base separation occurs, the point of detachment tends to move only a portion of the distance towards the leading edge and often results in only a minimal increase in drag. This is because the adverse pressure gradient associated with separation is smoothly eliminated as the slope of the ogive is reduced.

The above mentioned ogival boattails fit into a class of add-on devices that are highly dependent upon flow separation. In addition to those geometries, it was decided to investigate a second class of base add-on devices involving slant backs. Slant angles of 5° , 12.5° , and 30° were chosen for analysis and were scaled from work published by Ahmed, et al.² Figure 4 shows the geometry and dimensions of these add-on slants. These shapes are academically interesting since they involve complex boundary layer separation as well as freestream vortex interactions. Both the ogives and the slants add insight into flow management of the base of ground transportation vehicles.

COMPUTATIONAL FLUID DYNAMICS - To further assess the two ogive boattail geometries, the flow about the SANDRAG Vehicle with axisymmetric 5 ft and 8 ft ogival boattails was computed using RAMPANT. The predicted base axial force coefficient contributions (C_A , $\Psi = 0^\circ$) computed by SANDRAG and RAMPANT are listed in Table 1. For the present cases, SANDRAG predicted ΔC_A 's of 0.086 and 0.116, and RAMPANT predicted ΔC_A 's of 0.117 and 0.121, for the 5 ft and 8 ft configurations, respectively.

Figure 7 shows a RAMPANT velocity vector plot in the base region of the SANDRAG Vehicle with an attached 5 ft ogive boattail. It is clear that the wake takes the shape of the classical 'Low-Drag Hershey's Kiss'¹⁸. The point to be envisioned from this plot is that a truncated wake naturally takes the form of an ogive. This is a natural confirmation that an 'ogival' shape is near optimal for applications where maximum drag reduction is desired within length constraints.

STRENGTHS AND WEAKNESSES OF METHODS - The best approach for conceptual design proved to be a synergistic usage of multiple techniques. There is an extensive database of open literature available for general design. As the

need for parametric analysis arose, SANDRAG proved useful to show trends. The time to setup and compute a SANDRAG case was on the order of a few hours. Final analysis on a select few geometries could then be conducted with a resource intensive Navier-Stokes based code. Here, the time to setup and compute one of the two-dimensional axisymmetric cases was on the order of a few days. In depth discussion and presentation of the computational fluid dynamics GTS program and results are available from Reference 3.

EXPERIMENTATION

In support of the GTS program, Sandia conducted a 1:8 scale test in the Texas A&M University (TAMU) Low Speed Wind Tunnel. The primary objective of the test was to develop a database on the various GTS vehicle configurations for comparison with the results of the concurrent computational fluid dynamics study.

The closed circuit single return facility has a rectangular test section that is 2.1 m (7 ft) high, 3.0 m (10 ft) wide, and 3.7 m (12 ft) long. Near atmospheric static pressure is maintained within the test section and an external balance system recorded 6 components of force and moment. In depth discussion and presentation of the experimental GTS program and results are available from References 4 and 19.

MODEL HARDWARE - The 1:8 scale model, manufactured by Texas A&M, was installed 1.3 cm (0.5 in) off the test section floor. No boundary layer device (e.g., flat plate, moving belt) was utilized to enhance simulation of ground plane effects. Cylindrical wind fairings were installed around each of the four struts that served as model support, pressure and electrical conduit, and transfer of load to balance. Configurations included the GTS Baseline Vehicle (Figure 5), as well as both the 5 ft (Figure 2) and 8 ft (Figure 3) Ogive Boattails, and the three Slants (Figure 4) (5° , 12.5° , and 30°). Each configuration could be tested with wheels removed. Figure 8 shows the GTS model installed in the TAMU tunnel.

TEST - Testing was conducted over the yaw angle range of $\pm 14^\circ$ at a Reynolds number, Re_w , based upon trailer width, w , of 1.6×10^6 . In addition to standard force and moment data, and SAE Recommended Practice²⁰ wind-averaged drag coefficient data, extensive pressure data and flow visualization data were acquired. Up to 109 electronically scanned static pressure taps, concentrated upon the model symmetry plane and base region, were utilized to map the surface pressure contour. A seven hole probe was used to obtain three orthogonal velocity components and total pressure, at discrete points, in the wake region behind the model. General flow near the body was observed using a smoke wand. Surface flow on the body was visualized using yarn tufts taped to the body and a mixture of tempera paint and diesel fuel painted on the surface. Wake flows were visualized by positioning a yarn tuft grid fixed distances behind the model. 35 mm film and VHS video were utilized to record the test, including the flow visualization.

Tunnel blockage errors were judged by test facility personnel to be within the uncertainty of the correction capability. Thus, no blockage adjustments were made to the data.

STRENGTHS AND WEAKNESSES OF METHOD - Experimental wind tunnel testing can complement analytical and computational prediction and design techniques. It is espe-

cially useful and accurate for measuring the differences due to the relatively small effects of most add-on drag reduction devices. Although not conducted for the GTS program, other experimental test techniques can also be utilized in a complementary manner. These might include road testing, full scale coast down testing, water tunnel testing, and tow tank testing.

EXPERIMENTAL AND CFD RESULTS

A comparison table and plots are included for the computational fluid dynamics (CFD) results obtained from RAMPANT and wind tunnel results obtained from the Texas A&M Low Speed Wind Tunnel. Data are presented only where a CFD analysis was conducted. This included yaw angles of 0° and -10° for the GTS Baseline and the GTS vehicle with attached 5 ft Ogive Boattail, and included a yaw angle of 0° for the GTS vehicle with attached 8 ft Ogive Boattail. Due to time constraints, no CFD was conducted for a yaw angle of -10° for the GTS vehicle with attached 8 ft Ogive Boattail and no CFD was conducted for the GTS vehicle with Slants. All data in this section are for wheels-attached configurations.

FORCE AND MOMENT - Table 2 provides a listing of the six components of force and moment (C_N , C_Y , C_A , C_m , C_n , C_l). Agreement for yaw and roll plane coefficients (C_Y , C_n , C_l) was very good. Forces and moments in these planes, at the conditions evaluated, were relatively insensitive to complex separation and ground plane effects. The static yaw plane instability seen on the Baseline at -10° yaw is amplified with the addition of the 5 ft Ogive Boattail. The contour of the ogive produces a 'lifting' force in the yaw plane that acts in an unstable direction. However, in general, the restoring stability provided by the tire friction against the road surface easily compensates for the aerodynamic instability.

Agreement for pitch and axial plane coefficients (C_N , C_A , C_m) was not as favorable. Forces and moments in these planes were driven by the ground plane boundary layer as well as the free shear layer and vorticity associated with the trailer wake. Whereas the experimental testing was conducted without the benefit of a ground plane simulation enhancement device (e.g., moving belt), the CFD did simulate a moving ground surface. Of particular interest to the ground transportation industry is the body axis axial force coefficient. This is a measure of the resistance force that impedes forward motion along the axial direction of the vehicle. Relative to the experimental data, CFD tended to over predict C_A by a range of 0.07 to 0.15. Similar behavior was seen in two-dimensional test cases, and was resolved by increasing the grid mesh density in the regions of large viscous effects. For three-dimensional problems, this becomes increasingly more difficult due to the draw upon memory and computational resources. The CFD cases listed within Table 2 were computed on a single Sun Microsystems Sparc 10 Model 51 processor. The 0° yaw cases took one month to converge and required approximately 350 Mbytes of random access memory with a grid size on the order of 600,000 cells. The -10° yaw cases required double the amount of CPU time, memory and cells.

SYMMETRY PLANE PRESSURE COEFFICIENT - Figures 9 through 13 show a comparison of experimental and computational static pressure coefficient, C_p , as a function of

non-dimensional body axial station. Body upper and lower surfaces along the vertical symmetry plane (plane x-y in Figure 5) are included. In general terms, the plots show that the flow stagnates near the front grill of the tractor. On the upper surface it then smoothly expands over the cab and recovers near freestream conditions for the length of the vehicle. The lower surface sees an abrupt expansion at the sharp radius that simulates the bumper geometry. The pressure then recovers near freestream conditions, but deviations occur due to the venturi effect through the wheels and the ground effects. The GTS Baseline at yaw angles of 0° and -10° are shown in Figure 9 and Figure 10, respectively. Agreement is reasonable, except near the last 10-20% of the vehicle on the lower surface. The GTS vehicle with the attached 5 ft Ogive Boattail at yaw angles of 0° and -10° are shown in Figure 11 and Figure 12, respectively. Agreement here is reasonable throughout the contour, and is very good near the base. The GTS vehicle with the attached 8 ft Ogive Boattail at a yaw angle of 0° is shown in Figure 13. Again, even in the separated region of the lower surface of the ogives, the agreement is good.

HORIZONTAL PLANE PRESSURE COEFFICIENT - Figures 14 through 18 show static pressure coefficient, C_p , as a function of non-dimensional body axial station in a horizontal plane (parallel to plane x-z in Figure 5). For the GTS Baseline, this plane is located at the midpoint between the top of the trailer and underbelly, station $y/w=0.70$, and includes data from the front of the cab to the rear of the trailer. For the ogive boattails, this plane is located at the midpoint between the top of the boattail and the bottom of the boattail, station $y/w=0.89$, and includes data only on the boattails. The GTS Baseline at yaw angles of 0° and -10° are shown in Figure 14 and Figure 15, respectively. Agreement between the methods was excellent, as expected from the results of the side plane force and moment coefficients. The GTS vehicle with the attached 5 ft Ogive Boattail at yaw angles of 0° and -10° are shown in Figure 16 and Figure 17, respectively. The CFD tended to over predict the expansion on the curved ogive surface. The integrated effect of this in the axial plane helps to explain why the CFD predicted a higher axial force coefficient as compared to the experimental data. The GTS vehicle with the attached 8 ft Ogive Boattail at a yaw angle of 0° is shown in Figure 18. The agreement for this single test case was good.

BASE PRESSURE COEFFICIENT CONTOUR - Figures 19 through 23 show static pressure coefficient, C_p , contours in a projected base view, as seen from behind the trailer, looking forward. The CFD data are plotted on the left, and the interpolated experimental data are plotted on the right. A major fraction of the axial force coefficient is the integrated result of the difference in pressure between the forces on the front of the tractor and on the back of the trailer. On the assumption that there is minimal upstream influence from the add-on devices to the front of the trailer, it can be concluded that these base view contours tell a major part of the story as to their effect on 'drag'. The GTS Baseline at yaw angles of 0° and -10° are shown in Figure 19 and Figure 20, respectively. The GTS vehicle with the attached 5 ft Ogive Boattail at yaw angles of 0° and -10° are shown in Figure 21 and Figure 22, respectively. The GTS vehicle with the attached 8 ft Ogive Boattail at a yaw angle of 0° is shown in Figure 23. From the CFD results, it is apparent that in a global sense the GTS Baseline is the 'dark-

est', corresponding with the lowest C_p and thus the highest drag. This is followed by the GTS vehicle with the attached 5 ft Ogive Boattail. The 'lightest' base is clearly the GTS vehicle with the 8 ft Ogive Boattail, corresponding to the least negative C_p and thus the lowest drag. This is consistent with the trend seen in the Table 2 CFD axial force coefficient. From the experimental plots, it is more difficult to sense a difference in integrated, global, darkness between the GTS Baseline and the GTS vehicle with the attached 5 ft Ogive Boattail. However, it is apparent from the experimental contour plot for the GTS vehicle with the 8 ft Ogive Boattail that it has the smallest axial force coefficient. This is consistent with the Table 2 axial force experimental data, which predicted little difference between the Baseline and the 5 ft Ogive, but predicted a lower value for the 8 ft Ogive.

VELOCITY VECTOR - Figures 24 through 31 show the velocity projected in the y_{axis} - z_{axis} plane, V_{yz}/V_∞ , non-dimensionalized by freestream velocity. It is presented in planes aft of the base of the trailer with the CFD data plotted on the left, and the experimental data plotted on the right. The experimental wake pressure probe data was viable provided the total angle of attack of the flow, with respect to the freestream, was less than 170° . Thus, the experimental data plot on the right of each figure has erroneous data in regions of very large angle of attack (e.g., recirculating flow). In general this is obvious as noted by very large arrows, or regions where the arrows do not show a trend. There are methods available to experimentally measure flow at any angle; however, it was not feasible for the test due to time and cost constraints. CFD has the advantage that flow at any angle is contained within the solution, with the challenge being the post-processing effort involved in visualizing the three-dimensional images. The GTS Baseline at a yaw angle of 0° for axial distances behind the base of the trailer of $[x'/w: 0.35, 0.71, 1.06, 1.41]$ are shown in [Figure 24, Figure 25, Figure 26, Figure 27], respectively. Qualitatively, agreement is good between the CFD and experiment. It is apparent how the wake converges on the centerline of the trailer with increasing axial station (axial distance behind the trailer). Regions of recirculation, ground effect, and vorticity are apparent. The GTS vehicle with the attached 5 ft Ogive Boattail at a yaw angle of 0° for axial distances behind the base of the trailer of $[x'/w: 0.71, 1.41]$ are shown in [Figure 28, Figure 29], respectively. The flow is attached and very smooth on the upper and side surfaces of the ogive. The bottom flow structure is similar in nature to the GTS Baseline. The GTS vehicle with the attached 8 ft Ogive Boattail at a yaw angle of 0° for axial distances behind the base of the trailer of $[x'/w: 1.06, 1.41]$ are shown in [Figure 30, Figure 31], respectively. A flow structure similar to that of the 5 ft Ogive is seen.

SUMMARY

An overview was presented of the synergistic application of analytical, experimental and computational tools to the design of ground transportation vehicles. The focus of the research was to investigate the fundamentals of the base flow of a Class-8 van-type tractor trailer that would prove useful in fluid flow management.

1. A prime design constraint for maximum length exclusion for aerodynamic devices of 1.5 m (5 ft) was identified. Unintended consequences were listed that may compromise the effectiveness of the drag reduction device. The importance of a favorable life cycle cost study was argued.
2. Based upon a stated criteria, the commercial code RAMPANT was selected as the computational fluid dynamics solver for the study, along with associated solid model definition, grid generation, post process and presentation tools.
3. The Ground Transportation System (GTS) Baseline vehicle was established as the geometry for application of analytical, numerical, and experimental analysis.
4. The conceptual design process yielded five add-on devices for study. A class of add-on devices highly dependent upon flow separation included a 1.5 m (5 ft) long ogival boattail with a truncated base diameter of 1.8 m (6 ft), and a 2.4 m (8 ft) ogival boattail with a truncated base diameter of 1.5 m (5 ft). A second class of base add-on devices involving complex boundary layer separation as well as freestream vortex interactions included slant angles of 5° , 12.5° , and 30° scaled from work published by Ahmed, et al.²
5. A 1:8 scale test in the Texas A&M University (TAMU) Low Speed Wind Tunnel was conducted with the prime objective being development of an experimental database on the various GTS vehicle configurations. This proved useful for comparison with the concurrent computational fluid dynamics study as well as advancement of the aerodynamic literature for heavy ground transportation vehicles⁴.
6. RAMPANT computational fluid dynamic simulations were completed for several GTS configurations and comparisons were made with the experimental database. The comparisons were, in general, favorable; however, significant differences did occur for some conditions. The discrepancy in exact agreement is a function of limitations in both prediction methods. At 29 m/s (60 m.p.h.), the full scale GTS vehicle Reynolds number is 4.8×10^6 , whereas at 1:8 scale in the TAMU tunnel, Re_w was 1.6×10^6 . The experimental Reynolds number was limited by a) increased model and facility costs with an increase in model scale and b) increased compressibility effects with increased velocity. In addition, the experimental ground plane effects were not accurately simulated. Computationally, the simulation was limited by the fineness of the grid cells in the regions of the boundary layer, as well as the wake and the associated free shear layer. The CFD cases were run on a fully loaded modern scientific work station and required 400-700 Mbytes of random access memory and took 1-2 months to converge. In addition, state-of-the-art turbulence models are still only approximations to driving conditions.

ACKNOWLEDGEMENTS

Thanks are in order to: W. P. Wolfe, M. A. Walker and J. M. Nelsen for computational and analytical support; M. R. Garcia for transportation industry consulting; J. E. Suazo for experimental hardware design; J. Martinez and T. Selcer and the staff of the Texas A&M University Low Speed Wind Tunnel and Machining and Repair Facility for experimental hardware and testing.

Complementary work was conducted in a Cooperative Research and Development Agreement (CRADA SC94-1275S) with American Truck Advantage, Inc. The efforts of B. G. Ruefer, J. W. Hurt, and L. G. Marlow, Jr., were appreciated.

This work was supported under Sandia National Laboratories' Laboratory Directed Research and Development Proposal No. 94-S-00163.

REFERENCES

- ¹ Kennedy, J. L., K & K Enterprises, Personal Communication.
- ² Ahmed, S. R., Ramm, G., and Faltin, G., "Some Salient Features of the Time-Averaged Ground Vehicle Wake", SAE Paper No. 840300, 1984.
- ³ Hassan, B., Gutierrez, W. T., Wolfe, W. P., Walker, M. A., Ruefer, B. G., and Hurt, J. W., "Numerical Prediction of Aerodynamic Drag for Heavy Ground Transportation Vehicles", AIAA 95-1913, 13th AIAA Applied Aerodynamics Conference, June, 1995.
- ⁴ Croll, R. H., Gutierrez, W. T., Hassan, B., Suazo, J. E., and Riggins, A. L., "Experimental Investigation of the Ground Transportation Systems (GTS) Project for Heavy Vehicle Drag Reduction", SAE 960907 SP-1145, 1996 SAE International Congress and Exposition, February, 1996.
- ⁵ Federal Register, Volume 52, Number 49, Friday, March 13, 1987, Rules and Regulations.
- ⁶ Hurt, J. W., Attorney at Law, Dallas, Texas, Personal Communication.
- ⁷ Snizek, H., "Aerodynamic Fuel Saving Devices - Impact on Maintenance Task Force", Informal Handout of the S.11 Vehicle Energy Conservation Study Group, The Maintenance Council of The American Trucking Associations, 1994 Annual Meeting and Transportation Equipment Exhibition, February 28-March 4, 1994, Orlando, Florida.
- ⁸ RAMPANT User's Guide, Version 3.1, Fluent Inc., Lebanon, NH, 1994.
- ⁹ Pro/ENGINEER Modeling User's Guide, Version 13.0, Parametric Technology Corporation, Waltham, MA, 1994.
- ¹⁰ preBFC User's Guide, Version 4.3, Fluent Inc., Lebanon, NH, 1994.
- ¹¹ PATRAN User's Guide, Version 3.0, PDA Engineering, Costa Mesa, CA, 1994.
- ¹² TGrid User's Guide, Version 2.2, Fluent Inc., Lebanon, NH, 1994.
- ¹³ Tecplot User's Manual, Version 6, Amtec Engineering, Inc., Bellevue, WA, 1994.
- ¹⁴ Peterson, R. L., "Drag Reduction Obtained by the Addition of a Boattail to a Box Shaped Vehicle," NASA CR 163113, Aug. 1981.

¹⁵ Wolfe, W. P. and Oberkampf, W. L., "SANDRAG - A Computer Code for Predicting Drag of Bodies of Revolution at Zero Angle of Attack in Incompressible Flow", SAND85-0515, April 1985.

¹⁶ Wolfe, W. P. and Oberkampf, W. L., "Drag Predictions for Projectiles and Finned Bodies in Incompressible Flow," *Journal of Spacecraft and Rockets*, Vol. 24, No. 1, Jan.-Feb. 1987, pp. 14-22.

¹⁷ Cooper, K. R., "The Wind Tunnel Testing of Heavy Trucks to Reduce Fuel Consumption", SAE 821285, Truck and Bus Meeting and Exposition, Indianapolis, Indiana, November 8-11, 1982.

¹⁸ Smith, A. M. O., Stokes, T. R. Jr. and Lee, R. S., "Optimum Tail Shapes for Bodies of Revolution", *J. Hydronautics*, Vol. 15, Nos. 1-4, January-December, 1981.

¹⁹ Steen, G., "Subsonic Wind Tunnel Test of the Sandia National Laboratories Ground Transportation Vehicle", Texas Engineering Experiment Station Report No. TR-9514, The Texas A&M University System, College Station, Texas, July, 1995.

²⁰ "SAE Wind Tunnel Test Procedure for Trucks and Buses", SAE J1252 JUL81, SAE Recommended Practice, July 1981.

APPENDIX - Tables and Figures

Table 1. Axial force coefficient comparison between SANDRAG and RAMPANT

Configuration	Tail (ogive plus truncated base)	ΔC_A (Vehicle minus add-on; i.e., effect of add-on)
SANDRAG		
SANDRAG Vehicle	0.130	---
w/ 5 ft Boattail	0.044	0.086
w/ 8 ft Boattail	0.014	0.116
RAMPANT		
SANDRAG Vehicle	0.149	---
w/ 5 ft Boattail	0.032	0.117
w/ 8 ft Boattail	0.028	0.121

Table 2. Force and moment coefficient comparison between TAMU Wind Tunnel data and RAMPANT results

Configuration	Yaw Angle		C_N	C_Y	C_A	C_m	C_n	C_l
Baseline	0°	Experiment (CFD)	-0.01 (-0.16)	0.02 (0.00)	0.35 (0.47)	0.43 (1.07)	-0.07 (0.00)	0.01 (0.00)
	-10°	Experiment (CFD)	0.41 (0.16)	-1.45 (-1.49)	0.40 (0.55)	0.01 (0.70)	-0.36 (-0.15)	-1.22 (-1.16)
5 ft Ogive Boattail	0°	Experiment (CFD)	0.16 (0.10)	0.01 (0.00)	0.35 (0.42)	-0.40 (-0.44)	-0.06 (0.00)	0.01 (0.00)
	-10°	Experiment (CFD)	0.63 (0.47)	-1.41 (-1.43)	0.38 (0.48)	-1.05 (-0.45)	-0.57 (-0.45)	-1.19 (-1.07)
8 ft Ogive Boattail	0°	Experiment (CFD)	0.11 (0.04)	0.01 (0.00)	0.33 (0.41)	-0.21 (-0.09)	-0.06 (0.00)	0.01 (0.00)

Picture Courtesy of
Penske Racing.

Figure 1. GTS Baseline Vehicle

Figure 2. 5 ft Ogive Boattail including pressure tap locations

Figure 3. 8 ft Ogive Boattail including pressure tap locations

Figure 4. Add-on Slants including pressure tap locations

Figure 5. Dimensioned GTS Baseline Vehicle including
pressure tap locations

Figure 6. SANDRAG typical geometries

Figure 7. Velocity vectors from RAMPANT in base region of SANDRAG Vehicle with 5 ft ogive boattail

Figure 8. 1:8 scale GTS model in Texas A&M University Low Speed Wind Tunnel

Figure 9. GTS Baseline at 0° yaw, symmetry plane static pressure coefficient

Figure 10. GTS Baseline at -10° yaw, symmetry plane static pressure coefficient

Figure 11. GTS vehicle with 5 ft Ogive Boattail at 0° yaw, symmetry plane static pressure coefficient

Figure 12. GTS vehicle with 5 ft Ogive Boattail at -10° yaw, symmetry plane static pressure coefficient

Figure 15. GTS Baseline at -10° yaw, horizontal plane static pressure coefficient

Figure 13. GTS vehicle with 8 ft Ogive Boattail at 0° yaw, symmetry plane static pressure coefficient

Figure 16. GTS vehicle with 5 ft Ogive Boattail at 0° yaw, horizontal plane static pressure coefficient

Figure 14. GTS Baseline at 0° yaw, horizontal plane static pressure coefficient

Figure 17. GTS vehicle with 5 ft Ogive Boattail at -10° yaw, horizontal plane static pressure coefficient

Figure 18. GTS vehicle with 8 ft Ogive Boattail at 0° yaw, horizontal plane static pressure coefficient

Figure 19. GTS Baseline at 0° yaw, base static pressure coefficient contours (CFD left, Exp. right)

Figure 20. GTS Baseline at -10° yaw, base static pressure coefficient contours (CFD left, Exp. right)

Figure 21. GTS with 5 ft Ogive Boattail at 0° yaw, base static pressure coefficient contours (CFD left, Exp. right)

Figure 22. GTS with 5 ft Ogive Boattail at -10° yaw, base static pressure coefficient contours (CFD left, Exp. right)

Figure 23. GTS with 8 ft Ogive Boattail at 0° yaw, base static pressure coefficient contours (CFD left, Exp. right)

Figure 26. GTS Baseline at 0° yaw, V_{yz}/N_∞ velocity vectors at 3rd axial station ($x/w = 1.06$) (CFD left, Exp. right)

Figure 25. GTS Baseline at 0° yaw, V_{yz}/N_∞ velocity vectors at 2nd axial station ($x/w = 0.71$) (CFD left, Exp. right)

Figure 24. GTS Baseline at 0° yaw, V_{yz}/N_∞ velocity vectors at 1st axial station ($x/w = 0.35$) (CFD left, Exp. right)

Figure 27. GTS Baseline at 0° yaw, V_{yz}/V_∞ velocity vectors at 4th axial station ($x'/w = 1.41$) (CFD left, Exp. right)

Figure 28. GTS with 5 ft Ogive Boattail at 0° yaw, V_{yz}/V_∞ velocity vectors at 2nd axial station ($x'/w = 0.71$) (CFD left, Exp. right)

Figure 29. GTS with 5 ft Ogive Boattail at 0° yaw, V_{yz}/V_∞ velocity vectors at 4th axial station ($x'/w = 1.41$) (CFD left, Exp. right)

Figure 30. GTS with 8 ft Ogive Boattail at 0° yaw, V_{yz}/V_∞ velocity vectors at 3rd axial station ($x'/w = 1.06$) (CFD left, Exp. right)

Figure 31. GTS with 8 ft Ogive Boattail at 0° yaw, V_{yz}/V_∞ velocity vectors at 4th axial station ($x'/w = 1.41$) (CFD left, Exp. right)

DISCLAIMER

This report was prepared as an account of work sponsored by an agency of the United States Government. Neither the United States Government nor any agency thereof, nor any of their employees, makes any warranty, express or implied, or assumes any legal liability or responsibility for the accuracy, completeness, or usefulness of any information, apparatus, product, or process disclosed, or represents that its use would not infringe privately owned rights. Reference herein to any specific commercial product, process, or service by trade name, trademark, manufacturer, or otherwise does not necessarily constitute or imply its endorsement, recommendation, or favoring by the United States Government or any agency thereof. The views and opinions of authors expressed herein do not necessarily state or reflect those of the United States Government or any agency thereof.