

Boundary Interference of High Blockage Models in Open Jet Test Sections

Wael A. Mokhtar
Lake Superior State University

Colin P. Britcher
Old Dominion University

Copyright © 2008 SAE International

ABSTRACT

The boundary interference of high blockage models inside open jet test section is studied in three phases. First, a wind tunnel test was performed using a high blockage automotive model inside the 1/15th scale Langley Full Scale Wind tunnel. Second, a CFD simulation was done using CFL3D code (developed by NASA Langley Research Center). Finally, a panel method was used to assess the boundary interference and to study the effect of the collector. The objective of the study is to highlight the challenges in assessing the boundary interference for high blockage models. A secondary object is to present a model to integrate all the available information from the wind tunnel test and the CFD simulation to solve the problem using a panel method.

INTRODUCTION

Boundary interference is one of the main parameters that control the maximum allowable model size in wind tunnels. From the early thirties until the present day, methods have been developed to assess the amount of corrections due to this boundary interference [1] to [8]. The accuracy of the model and test section boundary representations has been the major differences between the methods. Interference assessment for open jet test sections is more challenging than with closed test sections because of factors such as the undefined boundary deformation and nozzle and collector constraint effects. Nearly none of the methods previously developed properly account for the boundary distortion in open jet test sections.

Essentially the same techniques used for closed jet test sections can be applied to the open jet case, albeit with a modified boundary condition (constant pressure).

However, classical method of images neglects the effects of jet boundary distortion, which have been shown to be significant in cases where the test model is large relative to the test section cross section. In other words, the correct boundary condition is enforced, but on an incorrect, undistorted boundary.

SAE has been active in reporting on the development of boundary correction methods for open jet automotive tunnels [9], [10]. Wickern described the use of the classical boundary correction methods for open jet and closed test sections during automotive tests [11]. In this application, the ground board represents the ground the vehicle is traveling on and has to be considered part of the tested model, not a test section boundary. To make the situation comparable to the one obtained in aeronautical applications, the model is mirrored at the ground plane and a situation with two models in a test section of twice the height (a duplex tunnel) is obtained. For this case all remaining boundaries are of the same nature (open jet or closed) and the classical method of images can be used. Mirror images are used to create the test section boundaries with the appropriate strengths to satisfy the boundary conditions on the test section boundaries. The blockage interference is then calculated by summing the streamwise velocity component induced by the image singularities. Wickern reported that the gradient correction, not accounted for in his method, has to be applied first before the obtained blockage interference [11]. It is important to note that the boundary deformation is not represented in this method. Also all the limitations of the classical method of images such as the assumption of a simple small model inside an infinitely long test section are imposed.

In addition to the above, the finite jet length, not properly accounted for in the classical theories, may be the source of the majority of the boundary interferences in an open jet wind tunnel. Mercker et al developed one of the

commonly used methods [12], [13]. They grouped the interference effects into blockage due to jet expansion, empty-tunnel pressure gradients, and nozzle or collector solid blockage effects. They modified the classical formulations to include the additional jet expansion and the additional velocity reduction at the model due to jet deflection caused by the model proximity to the nozzle. For the empty-tunnel pressure gradient they presented a simplified form instead of the full integration of the pressure over the model to obtain the horizontal buoyancy. For the solid blockage due to the nozzle and the collector they used a simple point source and two vortex rings positioned at the nozzle and the collector. The method is relatively efficient in correcting the drag force based on a single point correction approach, but it depends partially on the method of images, which may put some limits on its applicability to complex model geometries.

Several improvements of the above method were later presented. For example, Wickern et al presented a study of the nozzle gradient effect on open jet blockage interference for models of high blockage ratios [14]. The study consisted of three main parts: an analytical model, CFD simulation, and experimental validations. The analytical model was based on the momentum balance and potential flow theory. Its objective was to present a basic analysis of the dependencies of the effect of the model and the wind tunnel properties. A simple streamlined model was represented using a blockage doublet with its axis parallel to the test section centerline aiming toward the upstream direction. In the second step of the study a simplified two-dimensional CFD analysis was performed using the commercial code StarCDTM. A rectangular numerical domain was used. Its left boundary represents the nozzle exit section; its right boundary represents the collector entrance section, and the lower boundary represents the ground board. The model is merged to the lower boundary and the upper boundary represents the ceiling of the test section. The layout of the numerical domain used in this study resembles the one used in the present study except that the latter is extended to include the nozzle [14], the collector and the plenum chamber. StarCDTM solves the Reynolds-averaged Navier Stokes equations (RANS) and the k- ϵ model was used to simulate turbulence. The CFD-calculated values of the drag were corrected using the method developed in the analytical study and compared to a reference case, which was also obtained using CFD, but with 1% blockage. Finally the method was used to correct the effect of the nozzle pressure gradient for some of the published wind tunnel results and showed acceptable success.

Wiedemann et al extended the previous study of the nozzle pressure gradient to include the effect of the boundary layer pre-suction on the ground board upstream to the model [15]. They presented an experimental study conducted in the 1/5th scale IVK model wind tunnel in Stuttgart. The study showed that the boundary layer pre-suction appears as a ground simulation increment due to the decreased boundary

layer thickness and as a static pressure gradient increment due to the unintended additional pressure gradient. Based on the experimental results they proposed an empirical method to correct this effect by modifying the G-Factors (Glauert-Factor: a factor used to calculate effective volume of the model to correct the buoyancy effect of the empty test section pressure gradient) [1]. In their method they suggest slicing the model into sub-volumes and recommended the use of G-Factors in the order of one or smaller.

Recently, Mercker et al presented a semi-empirical approach to correct the blockage interference by correlating the changes in the measured drag to the pressure gradient over the wake [16]. The results from a test conducted in the model wind tunnel of IVK/FKFS of the University of Stuttgart were used to develop the method. An SAE generic vehicle model was used with four interchangeable rear-end shapes (square-back, hatchback, notchback, and fastback). The pressure gradient downstream of the model was changed by controlling the angle of collector flaps. The authors modified the method developed by Mercker et al [12], [13] where the model was represented by a point source singularity and the nozzle and the collector are represented by two vortex rings located at the nozzle exit section and the collector entrance. In the latter method [12], [13], the source strength was set to give a semi-infinite body of revolution with same base area as the frontal area of the vehicle. The source was located so that the leading edge of the semi-infinite body was at the front bumper location. This location, as they reported, was arbitrary in this method and they presented some recommended values based on the experimental study.

Hoffman et al developed a purely empirical method (linear regression) to assess the boundary interference of Ford/Sverdrup Drivability Test Facility (DTF), Wind Tunnel No. 8 [17], [18]. This wind tunnel is used for both aerodynamic and aero-acoustic tests and, as they reported, was in full operational status in 2001. In their first study [17], the objective was to compare two test section configurations: slotted and 3/4 open jet boundaries. Four MIRA (Motor Industry Research Association) automobile shapes and six SUV (Sport Utility Vehicle) shapes are used to represent blockage from 7 to 25%. The tests were conducted in the 1/11th scale representation of the Ford's Wind Tunnel No. 8. A reference set of data was obtained by testing the same models inside Sverdrup's sub-scale adaptive wall tunnel. No interference assessment was performed in this study [15] and the authors reported that the comparison study of the 3/4 open jet, slotted wall and reference results (adaptive wall) showed that the 3/4 open jet offers less aerodynamic interference. This coupled with the acoustic advantage of the open jet drove the decision to strictly employ an open jet configuration in the Ford's Wind Tunnel No. 8.

Later the same group (Hoffman et al [18]) extended the previous study to develop a purely empirical method to determine open jet lift and drag interferences for a wide range of vehicle shapes. The study included sedan, station wagon, pickup, and sport utility vehicle profiles for blockage ratio ranges from 11 to 25%. Again the tests were conducted in the 1/11th scale representation of Wind Tunnel No. 8 and the reference data was obtained from Sverdrup's sub-scale adaptive wall tunnel. They reported that the developed method showed effectiveness in correcting for drag interference while the lift correction did not work as well due to the measurement uncertainty. This method might be a simple and easy way to get an acceptable estimate of the boundary interference for the same models, test section and testing configurations. However, it is not a general method that can be safely used to correct other testing conditions. In addition, the method assumes that the reference data obtained from the adaptive wind tunnel is interference free which was proved to be not completely accurate as reported by Mokry [19].

More recently Mercker et al [20] presented an extension of the semi-empirical method to correct the empty-tunnel pressure distribution, described earlier [16]. The new method is also semi-empirical and it is based on wind tunnel measurements taken in the model wind tunnel of IVK/FKFS of the University of Stuttgart. The main improvement is that the new method calculated the drag correction by taking two wind tunnel measurements with different pressure gradients. One measurement was made in the standard wind tunnel flow. In the second one, the pressure gradient was introduced by closing the flaps of the collector. They reported that the new method was used to correct the drag using the following procedure. First two drag measurements were taken in two different pressure distributions. Second, based on the calculated static pressure difference between the front end of the vehicle and any downstream location, each of the measured drag coefficients was corrected using the method previously developed by the same authors. [12], [13] Then the two corrected drag coefficients obtained were compared. If they were identical, then the correction was found. If not, as they reported, the model had to be moved a small increment downstream and the previous steps were repeated. This work is currently the most up to date regarding boundary interference for automotive open jet wind tunnels. The method is presented by the same authors who developed the well known method described before.[12], [13] Again the method might be efficient in correcting the measurements it was based on; however, as they reported, it is semi-empirical and some of its steps depend at least partially on classical methods.

The literature cited above covers most of the previous and the current methods developed during the last decade for open jet boundary correction. The best description of the current status of the boundary interference correction methods for open jet test sections was presented recently by Mokry [21]. He reported, "To date, most corrections for open jet test sections are

either semi-empirical or reliant upon modified image-singularity solutions." In this study he proposed a method based on the potential flow formulations and a finite difference solver using successive over-relaxation. The method is computationally efficient, but as he indicated, the main weakness of the method is that the downstream area of the jet is assumed to have the same area as the nozzle area, so no account has been taken of the deformation of the tunnel jet boundary.

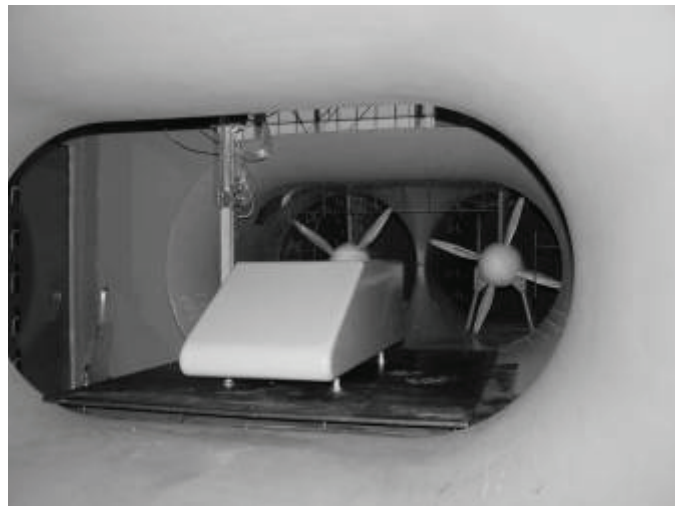


Figure 1: View of the "Davis" model inside the 1/15th scale LFST.

PRESENT WORK

A test was undertaken inside the 1/15th Scale Langley Full Scale wind Tunnel (LFST), where the main objectives were to apply a developed boundary correction method to a relatively high blockage ratio model configuration, to visualize the boundary deformation and to validate the computation of the boundary deformation. A secondary objective was an examination of the conditions at the flow interface to the collector. The geometry of the 1/15th scale LFST is very similar to the original (full-scale) wind tunnel. It has a 3/4 open test section with a 44.8 inch length, 24 inch height and 48 inch maximum width. A "Davis" [9], [10] generic truck model was specially manufactured for the test. Figure 1 shows a view of the model inside the test section. The model is 34 in maximum length, 9.6 in maximum height and 12.8 in width. The nominal blockage ratio is approximately 20.6%, which is a very high ratio for a wind tunnel test of external aerodynamics. The model's front face is 4 in downstream of the nozzle exit plane and its back face is approximately 7.3 in upstream of the collector entry plane.

The present work consists of three main sections. First, two sets of measurements are presented for the open jet boundaries with and without the model inside. The objective of this section is to visualize the boundary deformation in the test section region. Second, a simplified CFD simulation is presented to show more details about the behavior of the test section boundary (shear layer) and the jet interaction with the model.

Finally the boundary interference is assessed using a panel method developed by the current authors [22] to [28]. In this part, four approaches are used to model the wind tunnel. Each one of them presents a different level of approximation starting with a simple long test section with no collector, progressing to a fully deformed test section where the collector and the jet growth are represented. In this part, results from the CFD simulation are used in conjunction with the developed panel method.

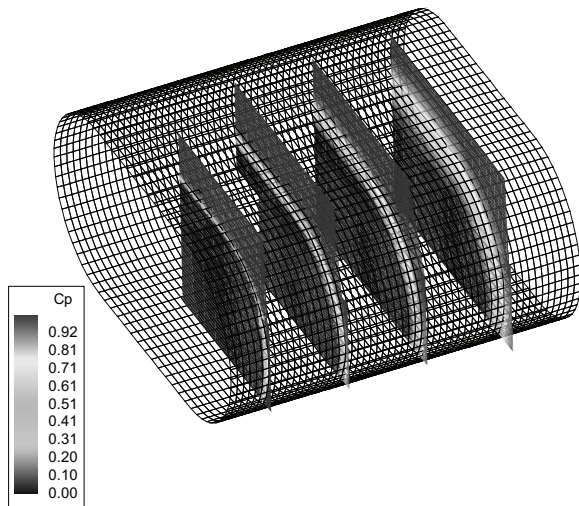


Figure 2: Velocity survey for an empty test section.

WIND TUNNEL MEASUREMENTS

A rake of 21 Pitot tubes, shown in Figure 1, was used to survey the test section boundaries in four vertical planes normal to the test section axis. Figures 2 and 3 show the velocity survey for an empty test section and for the test section with the model inside respectively. It should be noted that measurements are only available for the four planes depicted in Figure 2, with a measurement spacing of 1.0 inches in both the vertical and lateral directions. For both cases (with and without the model inside) the thickness of the shear layer at the test section boundaries increases in the downstream direction. This increase occurs at nearly the same rate for both cases, which suggests that the model does not have significant effect on the growth rate. Since the jet boundary is at practically constant pressure, this is thought to be reasonable. As the jet exits from the nozzle a thin mixing layer is developed around the jet boundaries, which grows in the downstream direction by entraining air from the plenum chamber. As the jet reaches the collector the entrained air is spilled back to the plenum chamber creating large vortices near the collector mouth as will be shown in the following section. Another view of the pressure contours for the measurements taken with the model inside is shown in Figure 4. Figure 5 is a sketch highlighting the important physical features of the flow field. It can be seen now that the boundaries of the test

section interact with the surrounding air creating this mixing layer, which makes it technically difficult to detect the effective test section boundaries and hence makes it more difficult to take accurate flow property measurements. These results will be used in the following section as a reference for the boundary deformation.

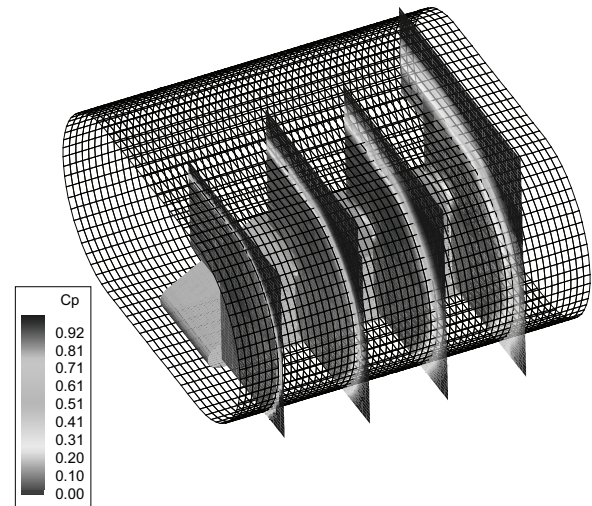


Figure 3: Velocity survey for the test section with the model inside.

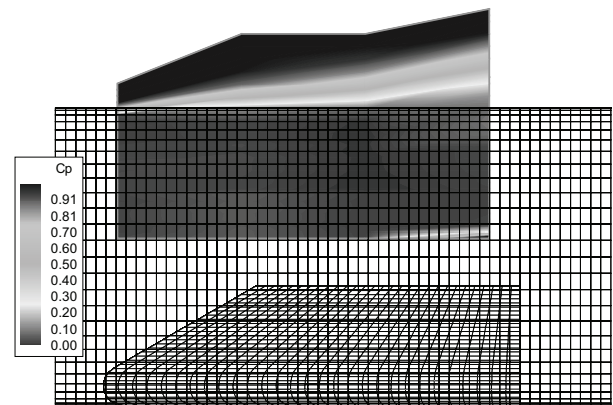


Figure 4: Velocity survey on a streamwise vertical plane passing through the test section centerline.

CFD SIMULATION

A 2D CFD study is performed using CFL3D. The principal objective of this study is to add more insight into the observations presented in the previous section through a full simulation of both the test section and the plenum chamber. Two cases are studied: the empty test section and the test section with the "Davis" model inside. The computational domain includes the nozzle, the test section, the collector and the plenum chamber. A two-block structured grid is generated for each case with clustering near to the nozzle, the collector surfaces, and