

Scania's New CD7 Climatic Wind Tunnel Facility for Heavy Trucks and Buses

Edward Duell, Amir Kharazi, and Paul Nagle
Jacobs

Per Elofsson, David Söderblom, and Christer Michael Ramden
Scania AB

ABSTRACT

Scania AB has opened the new CD7 climatic wind tunnel test facility, located at the Scania Technical Center in Södertälje, Sweden. This facility is designed for product development testing of heavy trucks and buses in a range of controllable environments. Having this unique test environment at the main development center enables Scania to test its vehicles in a controlled repeatable environment year round, improving lead times from design to production, producing higher quality and more reliable vehicles, and significantly improves the capability for large vehicle performance research.

This state-of-the-art facility provides environmental conditions from -35°C to 50°C with humidity control from 5 to 95 percent. The 13 m² nozzle wind tunnel can produce wind speeds up to 100 km/h. The dynamometer is rated at 800 kW for the rear axle and 150 kW for the front axle, which also has ±10° yaw capability.

In addition to the 'standard features' listed above for a climatic wind tunnel, this facility includes several additional capabilities. A Flow Control Device to improve the aerodynamic simulation around the sides of large, long vehicles in the test section was developed and implemented. Acoustic treatments in the circuit and test section plenum were designed to produce low background noise levels suitable for vehicle acoustic tests. This facility is also capable of producing complete simulation of solar, rain, snow, and soiling conditions. Moreover, emissions can be measured under road load conditions and in varying climates with different supplies of fuels.

This paper describes the engineering design of the climatic wind tunnel, important aerodynamic and thermal commissioning results, and performance validation results from initial tests with vehicles.

CITATION: Duell, E., Kharazi, A., Nagle, P., Elofsson, P. et al., "Scania's New CD7 Climatic Wind Tunnel Facility for Heavy Trucks and Buses," *SAE Int. J. Passeng. Cars - Mech. Syst.* 9(2):2016, doi:10.4271/2016-01-1614.

INTRODUCTION

Before the early 1990s, vehicle development engineers used "onroad" testing for vehicle development and design validation. Prototypes and development vehicles were tested on public roads or on specific test tracks. For climatic testing, the geographic locations for the tests needed to provide the required weather conditions (e.g. temperature, humidity, solar load) and terrain (e.g. road incline, altitude, and / or road conditions) for product development. Potential issues with "on-road" testing include dependency on natural weather conditions, traffic conditions for tests on public roads, travel time to get personnel and test vehicles to the remote location, availability of test engineers, and access to the limited number of development vehicles. Additionally, if either the vehicle or the test instrumentation failed during a test, attempting to return to the remote test location may be difficult due to development schedule and prototype availability constraints. Measurement results from on-road tests are also subject to higher variability, due to crosswinds, gusts, and atmospheric turbulence. For climatic testing, variations in temperature, humidity,

and solar load throughout a test day can add to the data variability, and therefore to uncertainty with design decisions [1]. Dedicated wind tunnels and other test facilities can overcome many of the issues identified here with "on-road" testing.

Therefore, since the early 1990s, there has been a worldwide automotive industry trend to incorporate test facilities as part of the corporate design center. For the automotive passenger car and truck industry, this trend towards increased vehicle development work in climatic wind tunnels has continued to grow as more test conditions can be simulated in a test facility. Examples of more specific test capabilities include altitude simulation, yaw testing in climatic wind tunnels, and rain, snow and soiling testing. Indeed, some global automotive firms have as many as ten climatic wind tunnels located in their design and development centers.

For the heavy truck and bus industry, there have been relatively few climatic wind tunnels [2] that have been purpose built with the dimensions needed and the dynamometer capabilities required. Scania

made the investment in a new climatic wind tunnel that is specifically designed only for heavy trucks and buses to provide this tool for development of their products [3]. Scania will now be less dependent on “on-road” testing and will be able to reduce product development schedules and to provide vehicles with better attributes for fuel consumption, emissions, driver comfort, and operational safety.

CD7 CLIMATIC WIND TUNNEL OVERVIEW

Jacobs was contracted to design and build a climatic wind tunnel for Scania AB in order to test heavy duty trucks and buses. The nozzle exit has dimensions of 3.7 m x 3.5 m (H x W), and a maximum test section velocity of 100 km/h. Detailed aerodynamic and thermal performance and other test capabilities are described below.

The basis for the design of the wind tunnel was developed jointly by Jacobs and Scania over a two year period. The wind tunnel circuit was laid out to have a minimum length and overall size in order to realize a cost effective building and facility. The relatively low maximum wind speed allowed a low contraction ratio which supported the compact size goal. Computational Fluid Dynamics (CFD) was used in the concept development phase to optimize certain wind tunnel component configurations.

The final circuit airline remained close to the circuit defined in the configuration development phase, with minor changes primarily to meet the velocity uniformity, turbulence specification and acoustic requirements. The final overall airline circuit in elevation is shown in [Figure 1](#) and the plan views are in [Figure 2](#). The individual components are pointed out in these figures. Discussions of the individual components follow below.

Development of the Wind Tunnel Circuit Airline

The wind tunnel airline was developed using traditional aerodynamic theory and guidelines and Computational Fluid Dynamics. For standard components such as turning vanes and contraction contours, traditional aerodynamic design theory and guidelines were utilized.

CFD served as a risk reduction tool to ensure that the aerodynamic design of certain wind tunnel components would work as intended. For components where the technical requirements were new and particularly challenging, CFD was used to evaluate different design concepts and the results were used to select the final geometry. Examples where CFD was used to develop new component geometries include the combined Corner 1 and collector configuration and the design and implementation of the Flow Control Devices (FCD).

CFD was also used to determine a velocity correction factor that was implemented into the wind tunnel airspeed control system to account for aerodynamic blockage effects associated with the trucks and buses tested.

CFD also provided freestream pressure coefficient (C_p) reference data (actual road data were not available) from the side of a bus for comparison to wind tunnel generated data. In addition, CFD results from a bus installed in the test section were used to design the size, shape and general location of the Flow Control Devices in the test section so that the pressure data from wind tunnel testing would have better agreement with freestream on-road C_p data.

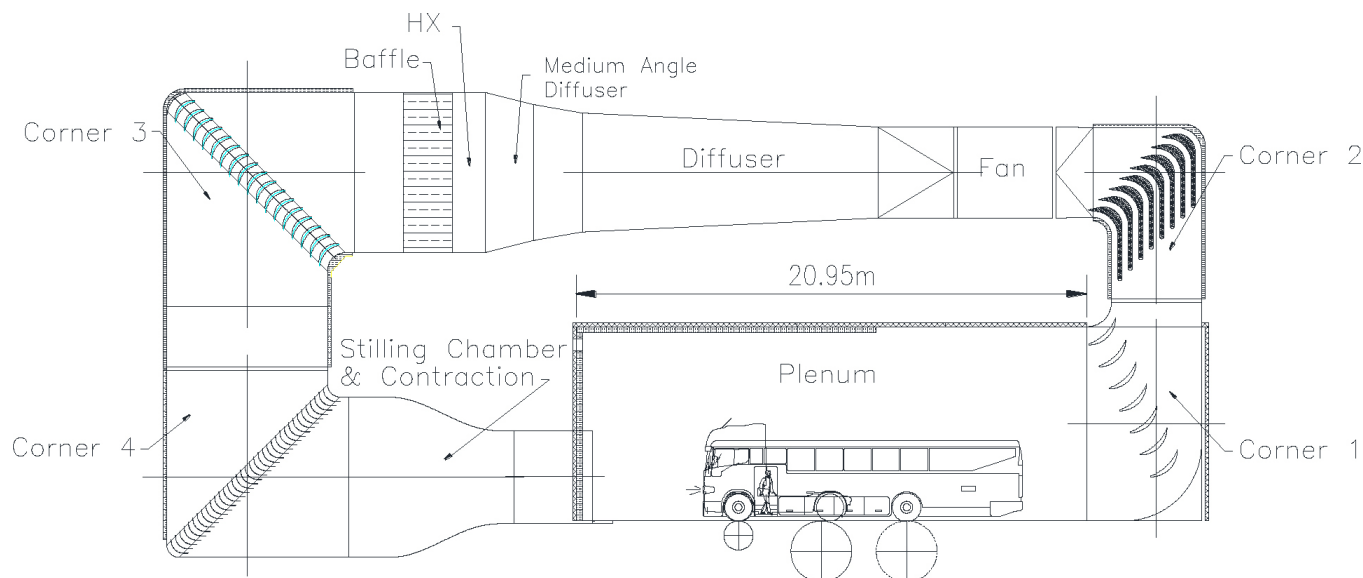


Figure 1. Scania airline circuit elevation view showing both truck and bus installed in test section plenum to indicate relative position.

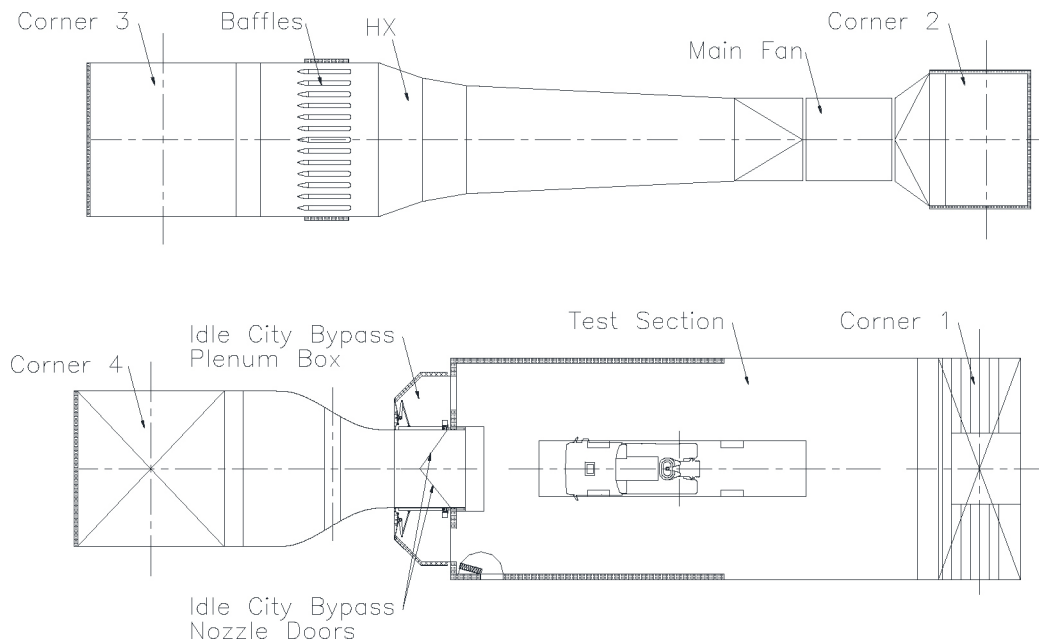


Figure 2. Scania airline circuit plan views of fan leg (top) and test leg (bottom) showing truck installed in test section plenum.

Flow Control Devices (FCD)

Accurate experimental testing in wind tunnels requires a similar flow field around the vehicles to that of the on-road condition. Providing ideal aerodynamic test conditions for large vehicles like buses and heavy trucks with frontal areas in the range from $\sim 8\text{m}^2$ to $\sim 11\text{m}^2$ is difficult, particularly when yaw testing is necessary. Following the traditional guidelines for aerodynamic blockage effects [4, 5, 6] in open jet wind tunnels would cause the nozzle exit size to become prohibitively large, from a cost perspective. It should be noted that the primary purposes of the Scania CWT include climatic, powertrain, aerodynamic and acoustic testing. The aerodynamic test objectives included flow field evaluation and soiling testing, and not overall vehicle force measurements. Therefore, the traditional aerodynamic blockage limitations concerning the ratio of vehicle frontal area to nozzle exit size were not strictly applied. Instead, to improve the local aerodynamic simulation with the restrictions of a small nozzle size, an innovative Flow Control Device (FCD) was designed and implemented in the test section.

The FCD system consists of two airfoils that can be positioned around test vehicles to create a local aerodynamic flow field that more closely resembles on-road conditions. One objective of the FCD is to modify the local flow field near the bus engine intake to simulate on-road conditions. The engine intake is located on the side of the bus, at the lower rear corner. The other objective is to modify the local flow field near the driver and passenger doors and windows of trucks at yaw conditions to simulate on-road conditions.

Scania specified the bus and truck geometry that the initial implementation of the FCD was to be designed for. Flexibility in positioning the FCD in the plenum allows the FCD to be used with other vehicles with different dimensions. The ideal position for other vehicle geometries would need to be defined with a CFD analysis process, as outlined below.

Creating a flow field in the wind tunnel that accurately represents the on-road flow near the engine inlet along the rear side of buses is important for development work. However, due to their large frontal area and length, when a bus is installed in a wind tunnel test section, the flow near the engine intake is decelerated, thus affecting the inflow into the engine inlet. To create a flow field in the wind tunnel that is closer to the on-road conditions near the engine intake, a Flow Control Device was designed with CFD tools and analysis.

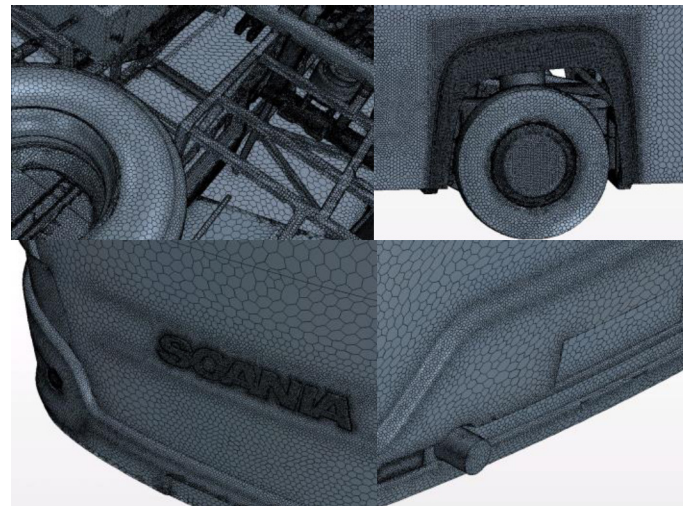


Figure 3. Detail surface geometry of the bus used for CFD analysis showing clockwise from upper left: underbody, wheelhouse, rear bumper, and front bumper corner details.

A detailed geometry of a typical Scania bus was used for the analysis, as shown in Figure 3. For the on-road “free air” CFD analysis, the bus model was inserted into a large rectangular box. The frontal area of the bus was 0.05% of the frontal area of the box to ensure that the free air mesh boundaries had no measurable influence on the aerodynamics around the vehicle. The commercial CFD package STARCCM+ v5.08 was used. Using surface wrapper technology and

a polyhedral mesher, a suitable grid resolution was designed. The free air model included over 30 million cells, starting with average 5mm cells at the surface and approaching 400mm at the free air wall boundaries. To accurately model the boundary layers, 12 prism layers with a 1mm layer at the surface and growth of 1.25 was used.

A steady state analysis was used with a standard k-e turbulence model and y+ wall treatment. Rotating tires and moving ground techniques were used to properly simulate the underbody flow field. Along the back of bus at the engine inlet grille location, flow was removed based on manufacturer's available flow rates and then reinjected from under body engine exhaust vents. Figure 4 shows CFD streamlines around the bus model from the free air case.

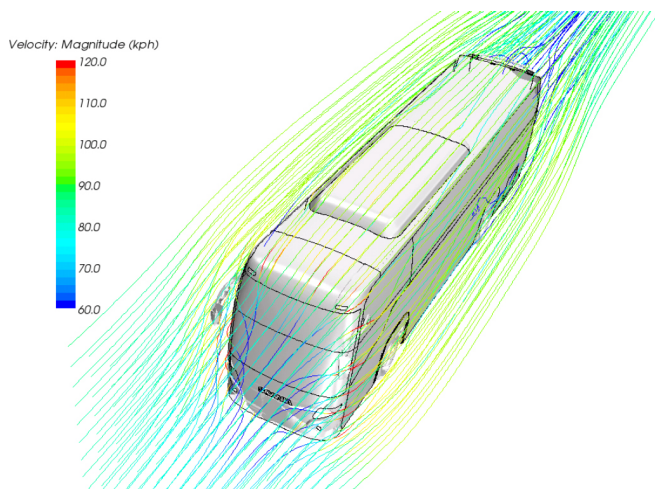


Figure 4. Flow streamlines around a Scania bus in free air simulation.

In the design of the FCD system, CFD was used extensively to determine the location and shape of the components. After the free air CFD analysis was performed, the bus model was placed in the test section plenum, 6 m from the nozzle exit plane. Similar mesh generation guidelines and the same physics were used as for the free air analysis. Figure 5 shows the CFD pressure distribution prediction on the surface of the bus in free air and in the baseline wind tunnel. The higher pressure coefficient along the back of the bus in the baseline wind tunnel demonstrates the need for an FCD system to improve the flow simulation along the backside of the vehicle.

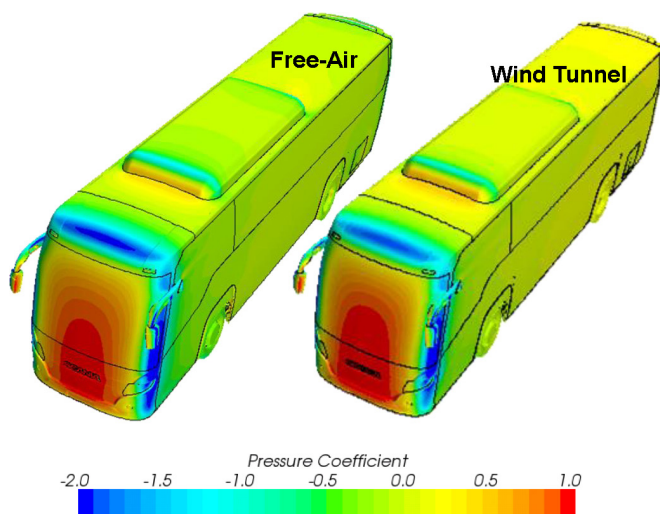


Figure 5. Pressure Coefficient on the surface of the bus in free air (left) versus in the baseline wind tunnel (right), without the FCD.

To create a flow field in the wind tunnel that is closer to the on-road conditions near the engine intake, the Flow Control Device developed with CFD was implemented. As shown in Figure 6, the airfoil shape of the FCD provides the necessary accelerated flow near the engine intake and then the flow diffuses downstream of the maximum airfoil thickness location on the FCD. Figure 7 shows a view of the bus in the test section with the FCDs located on each side.

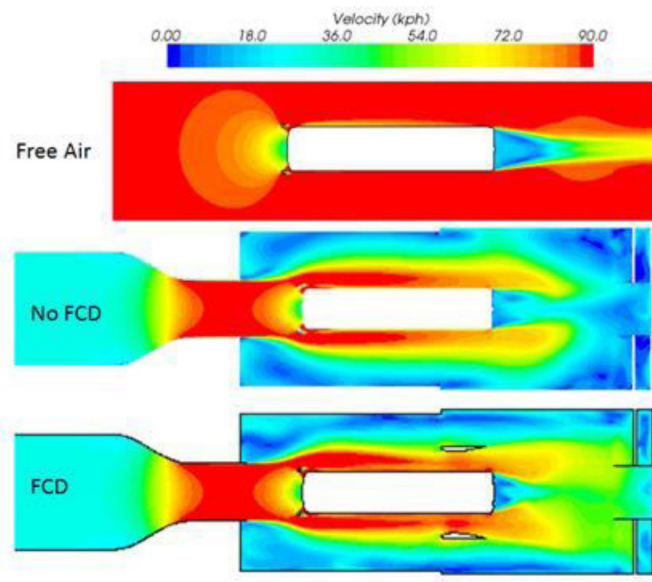


Figure 6. CFD results showing velocity magnitudes in a horizontal plane around a bus. Results shown are for on-road (Free Air) (top), with the bus positioned in the baseline CWT test section (center), and with the bus positioned in the CWT test section with FCDs (bottom).



Figure 7. Flow Control Devices (FCDs in black) positioned at 2.2m from the side walls near the rear of the bus. Pressure taps visible at $z = 2.5$ m. The bus engine inlet is visible at the lower rear corner, on the side of the bus.

Figure 8 depicts the aerodynamic requirement for the bus to match the FCD. The difference in pressure coefficient between the wind tunnel and on-road had to be less than ± 0.05 in the region of the

engine inlet (between $x = 6000$ and 8000mm , where $x = 0$ is at the center of the front axle). The CFD results from the development phase shows that the FCD successfully changed the pressure distribution in this region. Figure 9 shows measured results during acceptance testing of the wind tunnel with the FCDs in place. The measured data shows that the difference in pressure coefficient relative to the on-road CFD results was within the ± 0.05 limit.

It should be noted that CFD results were used as the reference onroad conditions. The options of full scale wind tunnel tests, scale model wind tunnel tests, and obtaining real on-road data were considered as sources for the reference data. Full scale testing even in the largest European wind tunnel [7] would result in blockage ratios of 10 - 15%, and significant data corrections would need to be applied. With scale model testing, there were concerns of Reynolds number effects on the pressure distribution along the side of the bus. It was decided not to use on-road tests due to the generally higher uncertainty of pressure coefficient data from repeatability, side winds, and ambient turbulence effects. While these technical concerns could certainly be accounted for, due to cost considerations and the schedule time required to organize full scale tests, it was agreed that the CFD results could be used as acceptable reference pressure data.

The other objective of the FCD system was to modify the local flow field around the truck cab during yaw testing. Due to the large blockage ratios of the truck cabs installed in the wind tunnel, the local flow field near the driver and passenger side doors and windows deviated from the on-road condition, in particular during yaw tests. During the development of soiling and wind-noise properties, it is important to have a correct flow field at the front corners of the truck.

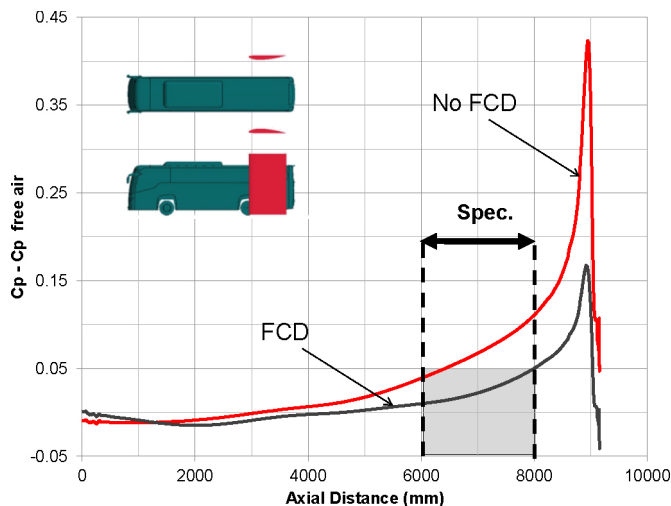


Figure 8. CFD results showing differences in C_p relative to on-road results on the side of a bus at 2.5m above the ground, with and without a Flow Control Device (FCD). $X = 0$ is at the center of the front axle.

To achieve this goal, a FCD was developed for yaw testing of trucks. The development method used CFD tools and analysis of the proposed configurations. The CFD methodology used for the truck FCD design was similar to that used for the bus analysis, outlined above. The FCD for this application involved re-positioning one of the FCD airfoils used from bus testing to the front of the test section and installing it in a “backward” position (trailing edge at the front).

Figure 10 is an off-axis view showing the FCD relative to the front of a truck cab at yaw. In this position, the FCD behaves less as an airfoil, and more like a flow deflector. Figure 11 and Figure 12 highlight the effectiveness of this device in improving the local flow field near the driver side window of the truck. The on-road (or free air) simulations show that the flow remains attached around the front corner of the truck, with a high suction peak (at $x = -1.1\text{m}$ in Figure 11) and high negative C_p downstream along the side window. The CFD simulations for the truck at 5 degree yaw with no FCD indicated local separated flow along the side window (center picture of Figure 12) and a reduced suction peak (at $x = -1.1\text{m}$ in Figure 11). The CFD simulations with the FCD in “backward” position shows that the FCD airfoil gradually redirects the flow around the front corner of the truck and then forces the flow toward the truck side window. This increases the local suction peak (Figure 11), and eliminates the local flow separation, as shown by the better agreement of the ‘Free Air’ and ‘Backward FCD’ simulations in Figure 12.

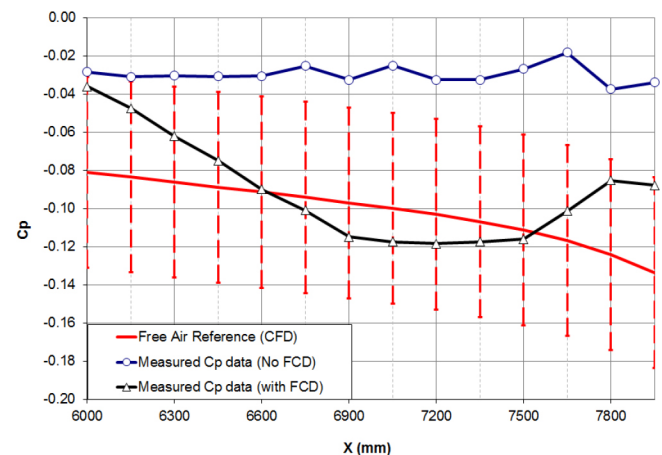


Figure 9. Influence of FCDs on measured C_p along the side of a bus at $z = 2.5\text{ m}$. ($x = 0$ is at the center of the front axle, and $z = 0$ is at the floor.). Vertical bars represent ± 0.05 allowable variation from Free Air reference C_p distribution.



Figure 10. FCD (black) on right side of photo in “backward” position at the front of a truck at yaw.