

Bluff Body Drag Reduction Using Passive Flow Control of Jet Boat Tail

Trevor Hirst, Chuanpeng Li, Yunchao Yang, Eric Brands, and Gecheng Zha
University of Miami

ABSTRACT

This paper conducts experimental study and numerical large eddy simulation for the drag reduction effect of jet boat-tail passive flow control on bluff body models. The jet boat-tail for bluff bodies operates by surrounding a converging duct around the end of a bluff body where the base surface is located. The duct captures free stream and forms a high speed jet angled toward the center of the bluff body base surface circumferentially to have the effect of a boat tail. A rectangular prism bluff body representative of various motor vehicle shapes such as trucks, vans, SUVs is used in this study. The numerical Large Eddy Simulation shows that the jet boat-tail sucks in the forebody boundary layer due to the low base pressure and significantly thins the boundary layer. The jet interacts with the shear layer and creates large vortex structures that entrain the freestream to base flow and energize it. The base pressure with the jet boat-tail is increased and the wake velocity deficit is reduced, resulting in a significant drag reduction. The Large Eddy Simulation indicates a significant drag reduction of 15%. The baseline and jet boat-tail configuration were also tested in a wind tunnel using 3D Stereo Particle Image Velocimetry at the speed of 10m/s and 30m/s. The wind tunnel testing shows a significant wake velocity deficit reduction by using jet boat-tail passive flow control, which is consistent with the drag reduction results of the Large Eddy Simulation.

CITATION: Hirst, T., Li, C., Yang, Y., Brands, E. et al., "Bluff Body Drag Reduction Using Passive Flow Control of Jet Boat Tail," *SAE Int. J. Commer. Veh.* 8(2):2015, doi:10.4271/2015-01-2891.

1. INTRODUCTION

Aerodynamic efficiency is at the forefront of concern when designing commercial road vehicles. As such, reducing aerodynamic drag has become the focal point of many research topics. Still, however, many commercial vehicles that are employed for the transportation of people and goods experience very high amounts of drag. Many of these high drag vehicles utilize configurations similar to rectangular prisms. Such vehicles include semi-trailer trucks, vans, buses, and SUV's. These vehicles are responsible for a substantial amount of miles traveled. Single-unit and combination trucks collectively consumed over 44 million gallons of gas in the US in 2010, accounting for over 26% of all gasoline consumed by motor vehicles that year[1].

A base surface is defined as a configuration ending with an abrupt cut-off by a flat or near flat surface [2]. The region immediately following the base surface is a volume that is of very low flow pressure and momentum, caused by this abrupt cut off. These characteristics make the base region a large source of drag, known as base drag. In fact, base drag is responsible for approximately 30% of all of the aerodynamic drag of a truck [2].

It is clear that reduction of this base drag would result in a noticeable reduction in the drag of the body, increasing the efficiency of the vehicle[1,2,3]. At a time when fuel consumption restrictions have become more and more stringent, base-drag reduction technologies have never been more crucial.

A plethora of base drag technologies have been explored for a variety of applications, including: base bleed, splitter wedges, splitter plates, and boat tailing [4, 5, 6]. These technologies are known as passive flow control methods. Passive flow control methods are methods in which they do not consume energy while reducing drag [7]. While these methods can be effective in reduction of drag, they are limited in their practical applications to semi-trailer trucks and other rectangular prism vehicles. Semi-trailer trucks face traffic restrictions limiting the available space for splitter plates and boat tailing [8]. Even though boat tailing is used for heavy-duty trucks with retractable panels. It is not so convenient to be used with smaller vehicles such as family vans, SUV's, cars, etc due to impeding the loading and unloading of a vehicle. In addition, base bleed requires a hole in the center of the base surface in order to allow for the formation of the counter-rotating vortex [9]. The location of such an open hole on a vehicle truck is usually unpermitted.

A concept known as Jet Boat Tail(JBT) has been previously suggested and proven effective to reduce the drag of automobile rear view mirrors by Zha and his team[10, 11,12,13,14]. This method is effective and convenient as it does not interfere with the base surface and does not create increase of the size of the body. In the previous study of the JBT flow control device, the inlet to the JBT is opened on the body of the rear view mirror model. This allows for airflow go through the model.

The purpose of this paper is to apply the passive Jet Boat Tail flow control to bluff body models, where the JBT is installed around the end part of the body and allows no flow to go through the prism itself.

2. JET BOAT TAIL CONCEPT

A conventional general bluff body model cross sectional sketch is illustrated in [Figure 1](#), which is observed to create vortex shedding and a large wake behind the bluff body.

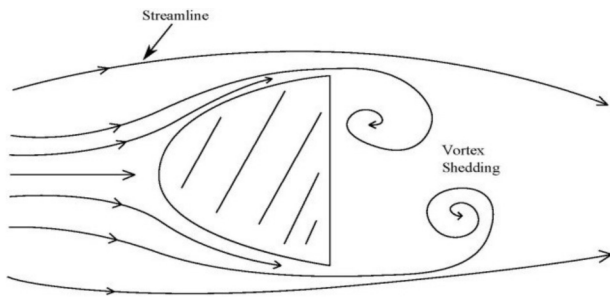


Figure 1. Vortex shedding flow structure of a general bluff body flow.

[Figure 2](#) is a cross section sketch of the bluff body model with the passive JBT flow control implemented. The flow control concept is to utilize an open inlet at the front leading edge of the configuration, introduce airflow via a converging duct, and eject the flow through a circumferential jet from the model base surface at an angle toward the center line of the wake region.

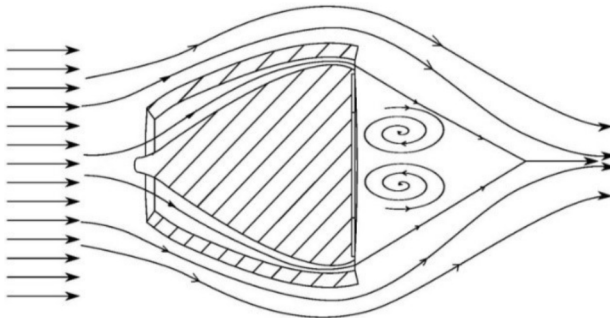


Figure 2. Expected flow structure with the jet boat tail effect influence.

2.1. Working Mechanism

The aerodynamic jet mirror working mechanism is the following [10, 11, 12, 13, 14]:

1. The jet harnesses the high kinetic energy by capturing a large amount of free stream flow with a large opened inlet in the front. It renders the jet to exit the surrounding of the mirror base with high kinetic energy and high total pressure.
2. The high energy jet creates a mixing with the main flow with large vortex structures, which entrain the main flow to the base flow and energize the base flow.
3. The angled jet toward the mirror center induces the flow to form a virtual “trailing edge” as illustrated in [Figure 2](#), creates a more stable vortex zone behind the mirror, mitigates the vortex shedding and turbulence fluctuation, and reduces the wake size. It is well known that the aerodynamic drag is directly determined by the wake width. The smaller the wake, the smaller the drag.
4. The energized base flow and shrunk wake size increase base static pressure. The opened inlet reduces the front blockage

by passing the flow and decrease the front area stagnation pressure area. These effects result in the reduced pressure drag. The reduced vortex shedding and turbulence fluctuation yields lower turbulence mixing noise that discomforts the driver and passengers.

3. THE JBT BLUFF BODY MODELS

To exhibit the benefits of the JBT passive flow control method for vehicles such as trucks, geometry models of rectangular prism with different rear end configurations are created for wind tunnel testing and large eddy simulation.

A sketch of the baseline model can be seen in [Figure 3](#). [Figure 4](#) shows the model with a Jet Boat Tail flow control device.

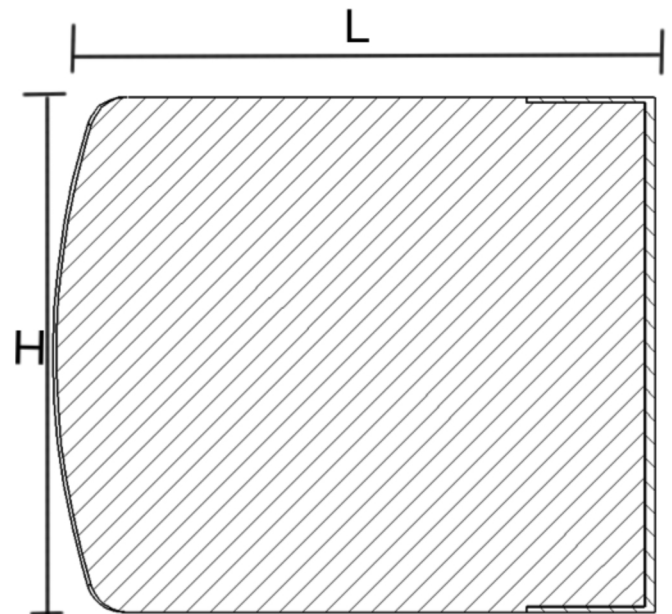


Figure 3. Cross Section of Baseline.

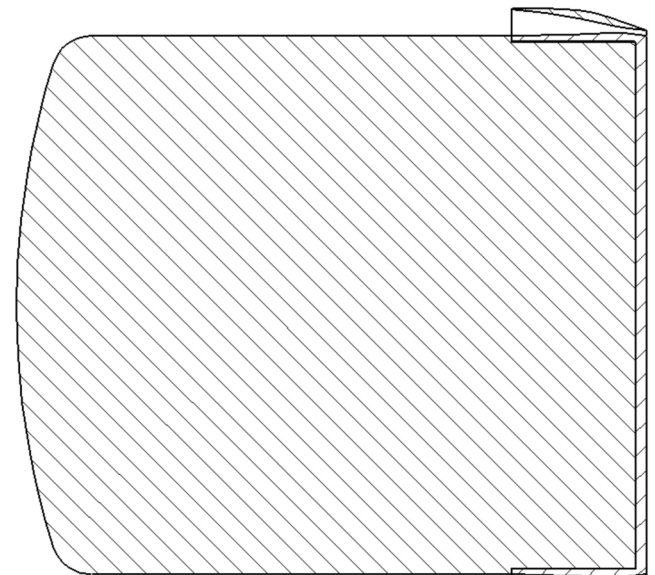


Figure 4. Cross section of model with JBT attachment.



Documents

Document Sharing Hub

The dimensions of the bluff body base, as seen in Figure 5, are height (H) of 200mm, a width of 150mm, and a length (L) of 230mm. The length includes a smoothly curved surface protruding upstream from the rectangular body to create the bluff characteristic.

The JBT configuration consisted of three separate pieces. These three pieces are referred to here as the bluff body base, inner shell, and outer shell. The configuration is illustrated in Figure 5.

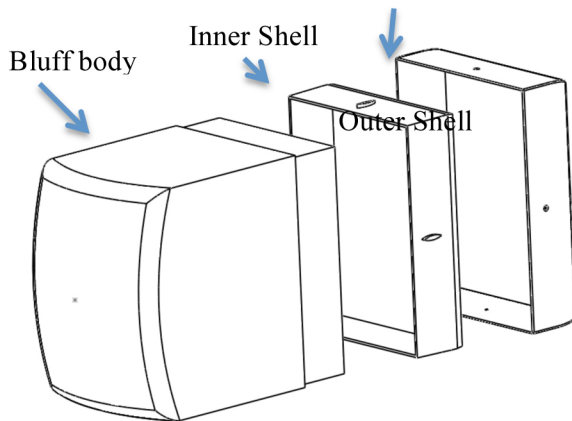


Figure 5. Exploded view of assembled model configuration.

The inlet cross sectional is uniform along all surfaces of the bluff body base, and is characterized by the gap between the inner and outer shell. The inlet height is 5% of the height of the bluff body base, which corresponds to 10mm. The outlet is similarly uniform along all surfaces, and has a height of 1%, or 2mm.

The duct length is 25%H, or 50mm. The duct profile, defined as the gap between the inner and outer shell is shown in Figure 6, converges smoothly from the inlet to the outlet. The flow exists the duct at an angle of 7° below freestream. The outer shell was defined to be as thin as possible, while still being able to maintain at high velocities, and be accurately 3D printed.

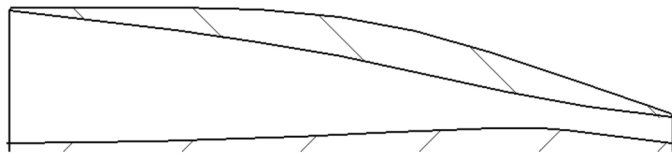


Figure 6. Profile of convergent jet section for model 10_2.

The baseline model consists of the bluff body base and an inner shell. The inner shell is created to fit smoothly and flat against the bluff body base.

4. PARTICLE IMAGE VELOCIMETRY(PIV)

All results presented here were obtained at the University of Miami Wind Tunnel Research facility. The wind tunnel used is an Open-Circuit Wind Tunnel, with test section dimensions of 24"×24"×24"

[15, 16]. The results were obtained using PIV experimentation. The model is mounted to an aluminum strut, which is attached to a load cell sting, such that the model is located in the center of the test section. Load cell data is not presented here, as the force measurements are not sensitive enough to accurately measure variations in the drag force.

A Lavision Digital Particle Velocimetry system is employed for the PIV experimentation. This system includes a Litron 135mJ Double Pulse Nd:YAG Laser and two Lavision proX2m series CCD Cameras. These Cameras are capable of recording up to 14Hz in a stereo PIV configuration. For each test, 400 samples were taken at approximately 6 Hz sample rate[17].

Table 1. PIV Instrumentation and Data Processing.

Target Flow of Measurement	
Target Flow of	2d/3C Air Flow in Wind
Uniform flow speed (m/s)	10m/s(Re=0.85x10 ⁵), 30m/s(Re=2.56x10 ⁵)
Tracer Particle	Di-Ethel Hexel Sebecate
Average Diameter	0.001
Light Source	Double Pulse Nd:TAG
Laser Power	800 mK
Laser Sheet Thickness	1.0mm
Time interval dt (μs)	78
Image Detection	
Camera	
Spatial Resolution	1600 x 1200 pixels
Sample Rate	~6 Hz
Pixel Size	.0074 mm x .0074 mm
Data Processing	
Pixel Unit Analysis	Stereo Cross Correlation
Correlation Area Size	64 x 64 pixels
Search Area Size	32 x 32 pixels
Integration Window	50% First Pass, 50%

An uncertainty analysis was performed in accordance with International Towing Tank Conference guidelines, resulting in a maximum uncertainty of 3.75%[18]. PIV velocity measurements of free stream were compared with Pitot tube results and found to be within 1%. The PIV plane measured for these tests was located 1/8th of the width of the model (18.75 mm) off of the center of the model as to avoid additional interference from the stabilizers.

5. RESULTS AND DISCUSSION

From the following aerodynamic drag equation

$$D = \int \rho u(u_{\infty} - u) dy$$

(1)