2004-01-0228

Experimental Investigation of the Flow Around a Generic SUV

Abdullah M. Al-Garni and Luis P. Bernal
Aerospace Engineering Department
University of Michigan

Bahram Khalighi
General Motors Corporation

ABSTRACT

The results of an experimental investigation of the flow in the near wake of a generic Sport Utility Vehicle (SUV) model are presented. The main goals of the study are to gain a better understanding of the external aerodynamics of SUVs, and to obtain a comprehensive experimental database that can be used as a benchmark to validate math-based CFD simulations for external aerodynamics. Data obtained in this study include the instantaneous and mean pressures, as well as mean velocities and turbulent quantities at various locations in the near wake. Mean pressure coefficients on the base of the SUV model vary from -0.23 to -0.1. The spectrum of the pressure coefficient fluctuation at the base of the model has a weak peak at a Strouhal number of 0.07. PIV measurements show a complex three-dimensional recirculation region behind the model of length approximately 1.2 times the width of the model. Turbulence properties are also reported, and the large-scale turbulent structure in the near wake is investigated using Proper Orthogonal Decomposition (POD) methods. The results suggest that the more energetic modes in the symmetry plane correspond to a vortex shedding process, and the more energetic modes in the center horizontal plane are a lateral flapping motion of the wake and a breathing mode of the mean recirculation region.

INTRODUCTION

In a recent investigation the flow in the near wake of a generic pickup truck model was investigated experimentally.[1,2] Mean and unsteady surface pressure, and PIV velocity data were obtained documenting the complexity of the flow in the near wake. Unsteady surface pressure measurements in the bed and tailgate showed a spectral peak at a non-dimensional frequency based on model width and free stream velocity of 0.07, and large fluctuation energy at low frequency. The former is due to the turbulent flow in the near wake, while the latter was attributed to the amplification of very small disturbances in the free stream. In that work, the development of the shear layers in the near wake was determined using PIV. The shear layers at the bottom and side edges of the tailgate are stronger than the cab shear layers. For the particular geometry tested, the cab shear layer does not reattach on the bed or the tailgate and extends to approximately the same downstream location as the tailgate shear layers. An interesting feature of the flow is the strong downwash in the symmetry plane of the model, which inhibits the formation of a mean recirculating region behind the tailgate. PIV measurements on planes normal to the free stream direction showed that the downwash is associated with compact streamwise vortices. However, the mean flow results show a counter rotating vortex pattern of size comparable to the width of the model. Similar flow structure in the wake of a pickup truck was reported by Leitz *et al*[3]. Bearman[4] reports PIV measurements of the streamwise vortices in the near wake of a car model showing similar features as found in the pickup truck flow.

In this paper we consider the flow around a generic SUV model. The front and underbody geometry of the SUV are the same as the pickup truck geometry investigated earlier.[1,2] The bed is covered to form a constant area section of the same cross section shape as the cab of the pickup truck, which is terminated in a blunt base. The model length is the same as the pickup truck length. The main objectives of the research are: 1) to gain a better understanding of the aerodynamics of SUVs including the flow structure in the near wake of the model, and 2) to obtain an extensive experimental data base for validation of computational models of these flows.

Particularly relevant to the present work are the investigations by Balkanyi et al[5,6,7] on the effects of drag reducing devices on the near wake of a bluff body with a blunt base. They used a model with a smooth nose designed to avoid boundary separation, a constant area section of rectangular cross section, and a blunt base where the drag reducing devices were attached. PIV measurements and unsteady surface pressure measurements showed that the drag reducing devices reduce turbulence intensity in the wake, while the shape