Experimental Thermal and Fluid Science 85 (2017) 344–353

Contents lists available at ScienceDirect

Experimental Thermal and Fluid Science

journal homepage: www.elsevier.com/locate/etfs

On the use of finite mixtures to improve the physical interpretation of a ground vortex flow

André R.R. Silva [a,*], Miguel R.O. Panão [b], Jorge M.M. Barata [a]

[a] AeroG-LAETA, Aerospace Sciences Department, University of Beira Interior, Rua Marques Ávila e Bolama, 6201-001 Covilhã, Portugal
[b] ADAI-LAETA, Mechanical Engineering Department, University of Coimbra, Rua Luis Reis Santos, 3030-788 Coimbra, Portugal

ARTICLE INFO

Article history:
Received 17 December 2015
Received in revised form 1 March 2017
Accepted 12 March 2017
Available online 16 March 2017

Keywords:
VSTOL
Ground vortex
Wall jet
Boundary layer
Statistical analysis
Finite mixtures
Bayesian approach
Markov Chain Monte Carlo

ABSTRACT

Laser-Doppler measurements of the velocity characteristics of a ground vortex flow resulting from the collision of a wall jet with a boundary layer are analyzed using advanced statistical tools, namely finite mixtures of probability density functions. These are determined by the best fitting to experimental results using a Bayesian approach based on a Markov Chain Monte Carlo (MCMC) algorithm. This approach takes into account eventual multimodality and heterogeneities in velocity field distributions. Therefore, it provides a more complete information about heterogeneous velocity distributions and its corresponding characteristic velocities and turbulent fluctuations. The ground vortex flow investigated is generated by a wall jet-to-boundary layer velocity ratio of 2. The results evidence how finite mixtures are able to reconstruct the measured probability distribution in the form of a mathematical probability density function. This allows to improve the physical interpretation of the ground vortex flow through quantifying its complex structure, which is particularly relevant to VSTOL aircraft flows. Namely, identify the separation point oscillation region, and the enlargement of the region comprising the effect of collision between wall jet and boundary layer in planes moving away from the wall. Also, in the collision zone, following a conventional statistical analysis, the *rms* velocity fluctuation (u') appears to be overestimated for the horizontal component due to the measured velocity range oscillating between positive and negative values. The results evidence how $\overline{U}$ and u' provide an idea of the flow dynamics, but their use is limited and an important amount of information associated with the highly curved flow complexity is lost. This prevents distinguish the magnitude of velocity fluctuations according to the flow direction, and the endorsement of anisotropy near the collision region, justifying the possibility of being numerically simulated.

1. Introduction

In nature, highly curved flows are common and frequently originated by impermeable surfaces that deflect a flow (e.g. Castro and Bradshaw [1]). The phenomena characterizing these complex flows include extra rates of strain and enhanced turbulence production through the interaction of normal stresses with normal strains, which is typical of impingement cooling applications in industry, and of the flow beneath short/vertical take-off (VSTOL) aircraft while lifting off or landing with zero or small forward momentum. In this latter application, the impingement of lift jets on the ground forms a wall jet propagating radially from the impinging point along the ground surface, interacting strongly with the ground plane, thus resulting in: lift losses; enhanced entrainment close to the ground (*suckdown*); engine thrust losses following re-ingestion of exhaust gases; and possible aerodynamic instabilities caused by fountain impingement on the aircraft underside. The interaction of this wall jet with the free stream forms a highly curved flow (ground vortex) far upstream of the impinging jet significantly influencing the flow development.

The literature only reports measurements for a secondary flow within the impinging jet configuration. In Barata and Durão [2], it is shown the shape, size and location of the ground vortex depends on the ratio between the jet exit and crossflow velocities, identified in two different regimes. One is characterized by the contact between the ground vortex and the impinging jet, while the other regime is detached upstream of the impinging zone. They also report a direct link between crossflow acceleration over the ground vortex and jet exit velocity, and an extended influence of the

* Corresponding author.
E-mail addresses: andre@ubi.pt (A.R.R. Silva), miguel.panao@dem.uc.pt (M.R.O. Panão), jbarata@ubi.pt (J.M.M. Barata).

http://dx.doi.org/10.1016/j.expthermflusci.2017.03.017
0894-1777/

upstream wall jet beyond the ground vortex, spreading upwards by a not well known mechanism.

Previous works [3–6] investigated Laser-Doppler measurements of velocity characteristics of two-dimensional ground vortex flows resulting from the collision of a wall jet with a boundary layer (Fig. 1) and discussed visualization results for wall jet to boundary layer velocity ratios (U_j/U_0) of 1.6, 1.7 and 2. A plane wall jet produced independently with a configuration previously used to study two-dimensional upwash flows avoids the influence of the impinging region [7]. The wall jet collides with the boundary layer produced by a conventional wind tunnel generating a ground vortex [3], allowing to investigate different velocity ratios between the wall jet and crossflow. Using the theory of turbulent jets and the distance to the separation point, it is possible to establish a relation between the wall jet velocity and the velocity at the jet exit. The results reported evidence for the first time the presence of a small vortex flow located upstream the separation point [8]. This secondary vortex has a low broadband pulsating behavior, expanding and contracting, as it is observed in some impinging jet configurations with ground vortex flows [9]. In a first stage, the tiny vortex grows, and the lower part of the boundary layer with anti-clockwise vorticity seems to merge into the growing vortex. In a second stage, as the small vortex continues to grow, it becomes larger than the boundary layer thickness, suddenly detaching, and convected upwards toward the curved flow. In a third stage, a new small vortex appears and grows, in a cyclic process restarting at stage one. Shear layer vortices are not at the origin of the secondary vortex growth, convected with the wall jet, since it cannot merge into the deflected flow resulting from the collision of the wall jet with the boundary layer. This is explained by the positive vertical velocity component above the vortex [3]. The unsteadiness of the ground vortex reported before for the case of impinging jets in unconfined crossflows may also be associated with an additional small vortex upstream the separation point, but, due to its small size, it is difficult to observe, particularly with high jet-to-crossflow velocity ratios [6]. The particular ordered sequence identified from visualization studies for the small recirculation zone near the separation point can also be interpreted as an oscillation of the separation zone or of the virtual deflected flow origin, and can be confirmed by the bimodal histogram of the horizontal velocity measurements made in this region [3]. In spite of the apparent organized sequence of the turbulent structure in the collision region, the power spectra of the horizontal velocity component exhibits no evident particular peak for the same location [3].

Barata et al. [10] presented a detailed analysis of the turbulent structure of a ground vortex flow resulting from the collision of a wall jet with a boundary layer, following the work reported in Refs. [3,4,7,11,12], having detected a small recirculating zone located upstream the separation. The authors performed a numerical simulation of the ground vortex upstream the stagnation point of an impinging jet under a crossflow. For a high jet-to-crossflow velocity ratio, relevant to V/STOL applications, they tested if the second (small) vortex forms due to a particular turbulent structure not yet analyzed or reported before. According to their hypothesis, a velocity ratio of 2.0 used between the wall jet and the crossflow corresponds to a regime where the small vortex is present. The conventional mean ($\overline{U}$) and turbulent (u) velocity components, and the Reynolds shear stress data are used to calculate the turbulent kinetic balances to understand the complex flow in the collision zone. In the collision zone of the wall jet with the boundary layer, turbulent kinetic energy balances show there is a local gain of energy by convection. Near the deflected flow, the convective term presents no significant contribution to the loss or gain of turbulent kinetic energy. Results further evidence, in the collision zone, the diffusive and dissipative terms, and the production term by shear stresses become predominant. The turbulent kinetic energy produced balances the loss by diffusion and dissipation. In the same zone, near the wall, turbulent kinetic energy is produced by convection, normal and shear stresses. The convective term is small and less than the production due to the normal and shear stresses. The collision zone between the wall jet and the boundary layer presents a behavior similar to a wall jet.

One of the main challenges in previous analyses is the loss of information when a mean value is calculated from a bimodal velocity distribution. Therefore, following previous works, the motivation is to explore advanced statistical tools, such as finite mixtures of probability distribution functions to better describe experimental velocity distributions, and improve the physical interpretation of two dimensional ground vortex flows resulting from the collision of a wall jet with a boundary layer. The section following the present introductory one is dedicated to the experimental method. Afterwards, the statistical method is described in the third section, before showing the results and corresponding discussion.

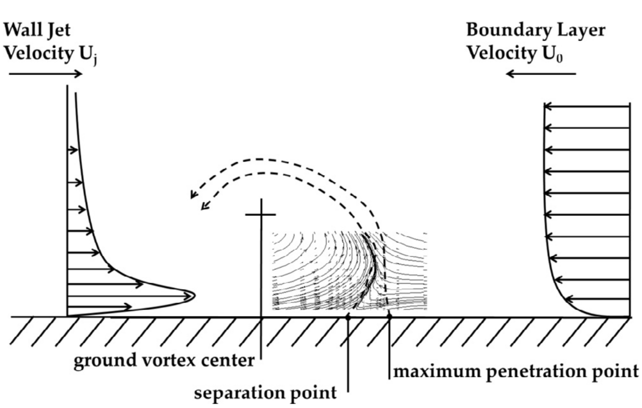

Fig. 1. Diagram of the ground vortex facility.

2. Experimental method

The wall jet collides with the boundary layer produced by a conventional wind tunnel, thus forming a ground vortex, which can be made of different velocity ratios between the wall jet and crossflow. In the present study, a smaller velocity ratio between the wall jet and the boundary layer of $U_R = U_j/U_0 = 2$ is considered.

Details of the experimental setup can be found in Barata et al. [12] and only a summary is given here. The wind tunnel facility is illustrated in Fig. 1 and shown in Fig. 2. The design followed the recommendations of Metha and Bradshaw [13] for open circuit wind tunnels, especially for the boundary layer part of the flow. A fan of 15 kW nominal power drives a maximum flow of 3000 m^3/h through the boundary layer and the wall jet tunnels of 300×400 mm and 15×400 mm exit sections, respectively.

The origin of the horizontal, X, and vertical, Y, coordinates is taken near the visual maximum penetration point. The X coordinate is positive in the wall jet flow direction and Y is positive upwards. Present results are measured at the vertical plane of symmetry for a wall jet mean velocity of 13.7 m/s and mean boundary layer velocity of 6.9 m/s, corresponding to a wall jet-to-crossflow velocity ratio, U_R, of 2. Considering the height of the tunnel as the characteristic dimension, the Reynolds for the boundary layer and wall jet are 1.4×10^6 and 1.5×10^4, respectively. Fig. 3 shows the wall-jet and boundary layer thicknesses at $X = -750$ mm and $X = 350$ mm, respectively.