8th. World Congress on Computational Mechanics (WCCM8)
5th. European Congress on Computational Methods in Applied Sciences and Engineering (ECCOMAS 2008)
June 30 – July 5, 2008
Venice, Italy

Large Eddy Simulation of Buoyant Jets for Airport Local Air Quality Studies

***Syoginus S. Aloysius[1] and Luiz C. Wrobel[2]**

[1] Brunel University
School of Engineering & Design
Uxbridge UB8 3PH
UK
syoginus.aloysius@brunel.ac.uk

[2] Brunel University
School of Engineering & Design
Uxbridge UB8 3PH
UK
luiz.wrobel@brunel.ac.uk

Key Words: *Buoyant Jets; Computational Fluid Dynamics; Large Eddy Simulation; Source Dynamics; Buoyancy Effects; Coanda Effect; ALAQS*

ABSTRACT

Air traffic movements are forecasted to increase from 5 to 7% in the near future [1]. If no measures are taken to control this expansion, the contribution to global warming from air transportation will raise about 4.3 times the current figures by 2050 [2].

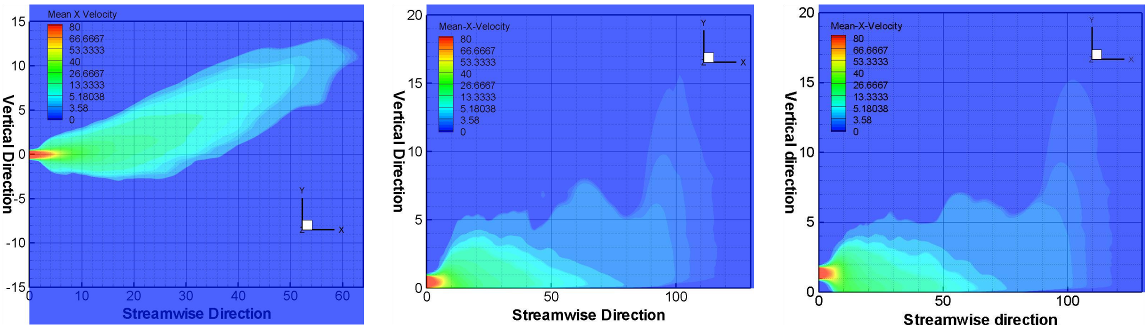
One of the uses of dispersion modelling techniques in the context of air traffic is to help operators and regulators assessing these impacts and quantifying the amount of pollution resulting from airport operations. Gaussian and Lagrangian models are the most commonly used techniques, which have been known to simplify considerably certain problems such as source dynamics. The main source contributors are the emissions from aircraft engines and from traffic inside and around airports [3]. These are all moving sources that need to be properly characterised in order to improve dispersion results. One of the most important parameters influencing the numerical simulations is the source dynamics of an aircraft taking off along the runway.

Computational Fluid Dynamics (CFD) is capable of providing increased knowledge and improved results that can benefit popular dispersion models to better represent the dynamics of an aircraft during the take-off stage. Before undertaking such difficult task, a staged process is necessary to understand the fluid mechanics aspects leading to a complete simulation.

The aim of this paper is to assess the impact of the presence of the ground on the flow properties by comparing the characteristics of a turbulent buoyant free jet, a turbulent buoyant wall jet and a buoyant jet at some intermediate distance above the ground.

Using Large Eddy Simulation (LES), a detailed investigation is performed on the transient characteristics of the airflow and pollutant dispersion. The basic concept of LES is to explicitly solve the larger eddies of the control volume as they contain most of the energy, do most of the transport of conservation properties, and are dependent on the geometry and the boundary conditions of the concerned flow. The smaller eddies are modelled through a filtering process due to their universality and their lesser influence on the fluid flow [4].

The numerical results of all the configurations are validated against existing experimental measurements of Forthmann [5] and Davis & Winarto [6], and some available analytical solutions from Tollmien [7], Goertler [7] and Zijnen's Gaussian Profile [8]. A comparison between the buoyant free and wall jets revealed several differences as shown in Figure 1. These differences will be discussed in detail in the paper.



Free Jet Simulation Wall Jet Simulation Above Ground Jet Simulation

Fig. 1 - Velocity profile comparison between different jets after 10s

First, the wall jet presents a longer potential core length than the free jet. This has an effect on the flow penetration through the control volume; there is a deeper penetration of the pollutant in the wall jet case. The maximum velocity decays much faster for the free jet than the wall jet, leading to the correlation between maximum velocity decay and plume penetration, as the penetration involves higher velocity pushing into the control volume.

The structure of the vortices is very similar for the free and wall jets near the exhaust, in the sense that counter-rotating vortices are formed, but their development is very different. The wall jet is continuously generating vortices due to the wall but, for the free jet, the influence of the vortices created by the surrounding fluid will gradually decrease. In the case of the wall jet, when the flow is progressing, the intensity of the vortices generated by the wall is much stronger than the one created by the surrounding fluid. This causes the flow to cling down to the wall for some distance; this effect is known as the Coanda Effect. In terms of fluid dynamics, there is a vertical restriction to the jet growth within a short distance behind the jet exhaust. As the velocity further away from the jet exhaust decreases, the wall vortices decrease and buoyancy takes over, lifting the flow from the ground.

When the jet is moved at some distance above the ground, results show a combination of both free and wall jet properties. Near the exhaust, the properties are very similar to the free jet whereas downwind the characteristics of the wall jet can be observed.

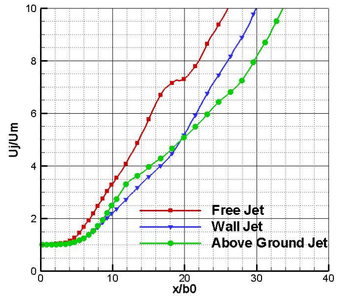


Fig. 2 - Velocity decay comparison between different jets after 10s

The decay of the maximum velocity comparison between different simulations presented in Figure 2 shows that the above ground jet has the lowest decay rate, in agreement with the findings of Davis & Winarto [6]. This has a consequence in terms of plume penetration; the above ground jet has deeper penetration of the exhaust gases through the control volume.

With an understanding of the relevant flow characteristics, it is possible to increase the complexity of the geometry to be simulated. In the context of take-off source dynamics, the complete geometry of an aircraft can now be modelled to study the interaction of different lifting devices on plume dispersion.

REFERENCES

[1] A. Graham and D. Raper, "Air Quality in Airport Approaches: Impact of Emissions Entrained by Vortices in Aircraft Wakes", http://www.cate.mmu.ac.uk/documents/Publications/Woct03.pdf, 2003.

[2] J.E. Penner, D.H. Lister, D.J. Griggs, D.J. Dokken and M. McFarland. "Aviation and the Global Atmosphere". http://www.grida.no/climate/ipcc/index.htm. Intergovernmental Panel on Climate Change special report.

[3] E. Fleuti, "Airport Local Air Quality Assessment", Eurocontrol report, Eurocontrol Experimental Centre, C20126E/DM/02, 2002.

[4] A. Bakker, "Modelling of Turbulence in Stirred Vessels using Large Eddy Simulation", Fluent Online CFD Book.

[5] E. Forthmann, "Turbulent Jet Expansion", English Translation, NACA Technical Memorandum TM-789, 1936.

[6] M.R. Davis and H. Winarto, "Jet Diffusion from a Circular Nozzle above a Solid Plane", J.F.M., Vol. 101, Part 1, pp. 201-222, 1980.

[7] G.N. Abramovich, "The Theory of Turbulent Jets", MIT Press, 1963.

[8] B.G. Van der Hegge Zijnen, "Measurements of the Velocity Distribution in a Plane Turbulent Jet of Air", App. Scientific Research, Vol. 7, pp. 256-276, 1958.