On the Use of Near Field Computational Fluid Dynamics for Improving Airport Related Dispersion Models

Syoginus S. Aloysius and Luiz C. Wrobel
School of Engineering and Design
Brunel University
Uxbridge, United Kingdom
Syoginus.Aloysius@brunel.ac.uk, Luiz.Wrobel@brunel.ac.uk

***Abstract*— This paper discusses one of the major problems concerning current dispersion modelling techniques used around airports; the source dynamics characterization. Due to the lack of information and non-availability of experimental data, common dispersion models rely on very simple source approximations. Through a staged process, the paper shows a more accurate representation of the plume dynamics of an aircraft during the take-off phase. Using Computational Fluid Dynamics, useful data can be collected to represent and understand the fluid mechanics associated with the dispersion process. The results can help dispersion modelers with better source dynamics representation and benefit the management of aircraft time separation delays in the take-off and landing phases.**

Keywords-Source Dynamics; Airports; Take-off; CFD; LIDAR; LES; Buoyant Jets; Ground Effects; Dispersion Models; Air Quality.

I. Introduction

There is a growing concern on the pollution resulting from airport operations because of the expansion of air traffic over the years. It is forecasted that future air traffic movements will increase at a mean annual rate of 5 to 7% [1]. According to Schafer et al., 15,000 aircraft are already populating the sky and an expected 2,200 billion of passenger kilometers are flown each year [2].

It is estimated that 3.5% of the global warming from human activity comes from air transportation, and it is predicted that this figure will rise to 15% by 2050 if no measures are taken to control air traffic, according to the intergovernmental panel on climate change [3].

Global effects such as depletion of the ozone layer and global warming are a direct consequence of local activities. To assess these impacts and quantify the amount of pollution resulting from airport operations, airport operators and regulators rely on different techniques to estimate existing situations and predict future scenarios. On-site monitoring is one technique that has been used but it is rather costly and does not isolate airport-related sources. Another option is the use of dispersion modelling techniques to approximate emission dispersion within a virtual domain. There are three modelling techniques commonly used in the aerospace industry, namely Gaussian, Lagrangian and Eulerian models. Each technique has advantages and drawbacks associated with the way they treat the problem. This paper examines one problem that has been known to both Gaussian and Lagrangian models, the source dynamics characterization.

It is known that the main source contributors at an airport are the emissions from aircraft engines and emissions from traffic inside and around the airport [4]. These are all moving sources, thus there is a real need of properly representing them in a simulation. This paper will first discuss why there is a need for improvement and how source dynamics are actually implemented in Gaussian and Lagrangian models. A discussion will then follow on different ways to characterize a moving source, before focusing on the real aim of the paper: the use of near-field Eulerian simulation to help improve Gaussian and Lagrangian models. To achieve this goal, several reports prepared as part of the ALAQS project (Airport Local Air Quality Studies), managed by EUROCONTROL, will be recalled and a staged process analysis will be done to finally analyze the dispersion process of a complete aircraft during take-off.

II. The Need for Improving Existing Airport Dispersion Models

There are three main types of dispersion modelling techniques, namely Gaussian, Lagrangian and Eulerian. The Gaussian model relies on a simple formula that calculates the concentration field emitted by a source “under stationary meteorological and emission conditions” [5]. In the Lagrangian particle models, the concentration is calculated through integration across the entire computational domain [5]. A gridding system is used to discretize the domain and to calculate localized concentrations within the grid. The Eulerian technique, on which Computational Fluid Dynamics (CFD) simulators are based, solves the governing equations of fluid flow with numerical methods. The basic idea of CFD is to discretize the control volume into small sub-domains, creating a grid system similar to the Lagrangian technique. The fundamental equations of fluid flow are solved either explicitly in the case of Direct Numerical Simulation (DNS) or are sub-

ISBN: 978-0-615-20720-9

modelled in the case of Large Eddy Simulation (LES) or Reynolds Averaged Navier-Stokes (RANS).

Fleuti [4] provided a simple comparison of these models in an airport context using the commercial software packages ADMS-Urban, LASPORT and EPISODE for Gaussian, Lagrangian and Eulerian models, respectively. The report shows that Lagrangian and Eulerian methods offer advantages over Gaussian models in both spatial and temporal resolutions [4].

Farias & ApSimon [6] studied the contribution of NOx from traffic and aircraft emissions around Heathrow airport. The Gaussian ADMS-Urban software was used to compare results obtained at different monitoring stations located in populated areas. They found some discrepancies between the two; the ADMS-Urban model overestimated aircraft contribution and underestimated the traffic contribution in comparison with monitoring data.

This over-prediction of airfield-related traffic was also observed by Fleuti & Hoffmann [7] in their study of Zurich airport with the Lagrangian model LASAT, in which simulation results were compared with monitoring sites located at different positions across the airfield. They concluded that the landing and take-off phase emission factors were lower in real condition than the simulated ones.

Fleuti et al. [8] also carried out a sensitivity analysis of Zurich airport, and pointed out that the emissions from the aircraft main engine dominate all other sources of pollutants. This was later confirmed by Celikel et al. [9] in their emission inventory at Zurich airport. Using the ALAQS-AV tool set, they showed that NOx emissions from aircraft sources around Zurich airport in 2003 were found to be approximately 80.7% of the total emissions [9].

III. Source dynamics treatment by Gaussian and Lagrangian models

Both ADMS and LASAT have integrated functions to take into account the dynamics of an airplane, but they are in a way very simplistic. The Gaussian ADMS model treats this problem by assigning an accelerating jet source to represent the effects of buoyancy and momentum of an aircraft engine [10]. Farias and ApSimon [6] criticized this approach because it is lacking features that “may contribute to thermal buoyancy” for the traffic emissions. In the case of aircraft emissions, the pollutant release occurs at different heights with fast moving sources during a short period of time [6]. They later added that ADMS, although highly suitable for dispersion from stacks, needs adjustments for ground traffic use because there are differences in “mechanics governing dispersion from these emissions” [6].

LASAT, on the other hand, rely on parameters calibrated by means of Dusseldorf DOAS measurements, which are also available for APU, GPU and vehicle source dynamics [11]. To characterize the take-off phase, LASPORT used an initial uniform box of 50m wide and 25m high having parameters (velocity and turbulence) of the exhaust plume decaying in a time scale of a minute or two [12]. Fleuti et al. [8] pointed out a major drawback concerning this method, because these parameters are not fixed and rely on constant changes to get the adequate results. For instance, they needed to increase the vertical source extent for vehicle exhaust they studied from 2m to 8m, and the Auxiliary Power Unit (APU) was also changed using more recent information from the Frankfurt airport air quality studies [8].

The ALAQS-AV tool set has the capability to impose a smooth and shift approach on these different modelling techniques [11]. This concept is based on the principle that emissions from engine exhausts can be shifted spatially downwind and smoothed to replicate the real plume dynamics of the jet engine. The smooth and shift approach used by ALAQS-AV employs LASPORT default parameterization values for different types of moving sources present at the airport [13].

Because of its time and length scales, the emission is done through an hourly passive grid source. A major drawback of this technique is its non-accountability of the meteorological conditions; the effects of the wind direction and magnitude are not properly treated, and this has a consequence in the directional exit of the plume and its rise [13]. These parameters are set in the model without any differentiation of the type of aircraft which was operational within the hour. This “one type set of data fits all” approach has also been criticized by Farias and ApSimon [6] because ADMS uses only one set of buoyancy and momentum parameters for all aircraft.

Yu et al. [14] showed that the major disadvantage of using air quality models is the lack of knowledge on emissions rates of pollutants from different types of engines, and these vary greatly during different phases of an aircraft. This argument was also raised by Schafer et al. [2], who pointed out that the only source of data available up to now is the ICAO database, but unfortunately this does not give any information whatsoever about the source dynamics. Fleuti et al. [8] also expressed the need for information concerning the plume characteristics in their sensitivity analysis; this parameter was found to have a major influence in the dispersion process. Like Schafer .et al. [2], they argued that the knowledge of how much emission is being released is important but the dynamics associated with it is also of great relevance.

IV. Inadequacy of Lidar measurements for source dynamics characterisation

One possible answer is to use Light Detection and Ranging (Lidar) equipment. Lidar systems consist of the transmission of a light signal pulse through a laser sheet; particles emitted from an engine backscatter the light and the laser receives back the signal, showing the presence of a certain particle at a location along the sheet [15]. Depending on the quality of the equipment, it is possible to analyze the results up to every second, with a resolution of about 10 meters, operating at the ultraviolet wavelength of 355nm [16], but this comes at a high price as a complete mobile facility has to be built. Examples are the Rapid Scanning Lidar Facility (RASCAL) introduced by the Manchester Metropolitan University or the Ozone Profiling Atmospheric Lidar (OPAL) by the National Oceanic and Atmospheric Administration. RASCAL is a self-contained mobile unit with the necessary equipment to work autonomously with on-board meteorological facility. Apart

 ISBN: 978-0-615-20720-9

from the equipment cost, there are several disadvantages associated with the use of Lidar to characterize source dynamics.

Starting with the meteorological conditions, it is likely that the wind speed and direction will not be constant and the changes that may occur will alter the fluid dynamics, hence dispersion. This makes it harder to compare one observation to another and correlate the results in order to have a proper understanding of the dynamics.

To be able to capture a pollutant, the particle has to reflect. In other words, the size of the particle is important. Usually aerosol, in the order of "100 nanometers with an average of 30 nanometers", is measured to give an approximate description of the plume on the laser sheet [15]. Wayson et al. [15] also found that these particles can only be seen using their OPAL system operating in the very low ultra-violet wavelength (355nm). This means that this device has to be used to its limit to capture the parameters of interest.

This search for small particles causes some problems, especially around airports where several other sources can pollute the Lidar results. Eberhard et al. [16] reported some of the problems concerning the low contrast between the backscattering and the signal from the ambient air. They found that some aircraft presented very low or no plume relative to the ambient air, and explained that this was due to backscatter particles having the same size as the ambient air. What is more intriguing is the fact that they found conditions where the plume has less backscatter than the ambient air, due to "a combination of volatilization of the ambient particles passing through the engine and low particle emissions" [16]. Another possible explanation is that the background concentration and interaction with other sources present around the airport are greater than the emission released by the engines.

According to Angus Graham from Manchester Metropolitan University (private communication), Lidar data have to be processed in order to be readable, and this involves a considerable amount of processing and manpower, at a cost that cannot be neglected. Eberhard et al. [16] also pointed out that only 40% of their measurements were retained after this "processing and quality control". This finally gave the plume characteristics of only 21 types of aircraft.

To conclude this discussion on Lidar, only a sheet can be analyzed making the analysis two-dimensional. It is often interesting to know the dispersion process not only on the vertical plane but also in the lateral direction, because some concentration may be trapped into wing-tip vortices and travel around the airport, as will be shown later in this paper. Another problem with the analysis is that it is only qualitative. Moreover, there are disagreements concerning the take-off phase. Wayson et al. [15] found compact ground-based plumes for all types of airplanes, whereas Graham et al. [17] in the Project for the Sustainable Development of Heathrow (PSDH) report found a non-compact ground plume for the B747-400. Yamartino et al. [12] tried to directly compare LASPORT's initial box with the plume size dimensions reported by Wayson et al. [15], and found it rather impossible to do so because of the time scale used by both analyses. Ignoring this, it was revealed that LASPORT produced values twice as large as the ones reported by Wayson et al. The explanation for this difference is that the measurement values taken by Wayson et al. are for the plume at a very early stage, before even the dissipation of the plume's internal and thermal energy had taken place [12]. Yamartino et al. [12] concluded that issues of plume entrainment and rise characterization have to be resolved before Gaussian or Lagrangian models can perform accurately in airport-related dispersion studies.

V. EULERIAN MODEL AS A WAY TO UNDERSTAND AND CHARACTERIZE SOURCE DYNAMICS

As discussed previously, Fleuti [4] compared the results of the Eulerian commercial package EPISODE to ADMS-Urban and LASPORT. This is a unique comparison between the three techniques, but unfortunately EPISODE is not a 100% Eulerian model. The calculation of the large scale dispersion process is done through an Eulerian grid where an averaged form of the Navier-Stokes equation is computed via the K-theory [4]. The small scales, on the other hand, are calculated through a subgrid scale Lagrangian or Gaussian model [18]. These near source treatments are usually done by point or line source dispersion [18].

Within the ALAQS project initiated in 2003 by the Eurocontrol Experimental Centre, Aloysius et al. [19] provided a comparison between CFD and LASAT simulations for airfield emission dispersion. It is believed this report was the first attempt to compare full Eulerian and Lagrangian models for pollutant dispersion around an airport. The CFD studies were based on the Large Eddy Simulation (LES) method to predict the dispersion of NOx, assumed to be a non-reactive pollutant, from the airport runways. A number of other simplifications were introduced to the model to reduce computing time, such as the terrain and the runways which were respectively flat and modelled as area sources.

Although the simulation of the air traffic at Zurich airport was done for a one-day period only, the results for the fourth hour of the day were interesting because of the changes in wind direction and magnitude, and emissions values, that happen during that period of time. Those changes were found to happen four times during that particular day, mainly due to airport operations or severe meteorological conditions.

The predicted dispersion patterns and the magnitudes of the NOx concentrations showed good agreement between the Eulerian and Lagrangian models. The notable differences were that LASAT predicts higher concentrations at ground level than CFD. At higher altitudes, LASAT predicts an intuitive dispersion throughout the control volume whereas CFD presents some recirculation. Although the magnitude of this recirculation is in the order of less than 1 ppb in this case, this can be higher for larger airports. In addition, it was found with the CFD simulation that:

- Vortices play an important role at high wind speeds, making the flow recirculate around the control volume.
- Buoyancy is dominant when emissions are high and wind speed relatively low.

ISBN: 978-0-615-20720-9

- Large vortices contain high concentration of pollutant and do most of the transport.
- The influences of small eddies are small, and their number increases when large vortices break down due to the surrounding flow and the wind magnitude.
- Changes in wind direction alter the flow rotation creating vortices in different directions.

It has also become apparent from this study that the application of CFD to the simulation of a full airfield is currently unreasonable. The CFD model required more than 8 days of processing time to reach its transient solution for a single hour simulation using 8 dual core processors optimized for parallel processing. In comparison, LASAT took about 60 seconds to compute the solution. But it was concluded from this report that CFD allows for a much in-depth assessment of how emissions are transported, and the wealth of information it provides means that it would be extremely beneficial if applied to more localized airfield studies such as source dynamics.

The strategy adopted in this study to validate the CFD results was to gradually increase, through a staged process, the complexity of the simulation towards representing the near-field effects of aircraft exhaust plumes under realistic conditions. This paper will go through this process from a free jet engine to a complete aircraft on the runway.

The LES model was initially used to investigate the differences between turbulent buoyant and non-buoyant jets in a free atmosphere condition, highlighting the mechanism of dispersion behind the exhaust. The non-buoyant free jet was compared with existing experiments and analytical results [20]. The non-buoyant simulation results were found to agree with classical results, replicating the characteristic self-preserving behavior of a free jet after the flow development region (Fig.1). The buoyant jet, on the other hand, breaks the symmetrical pattern of the flow showing a rise of the axial velocity above the centerline axis, as illustrated in Fig. 2.

The disappearance of the potential core was found to be the result of turbulence penetration, leading to the fully developed region. In the potential core, the jet entrains the ambient fluid surrounding it, triggering an enhancement of turbulence. Its intensity begins to act and increases further along the axis, resulting in a decay of the axial velocity. Compared to the buoyant jet, the non-buoyant jet has a longer potential core because it is not restrained by the buoyancy effects acting on the flow.

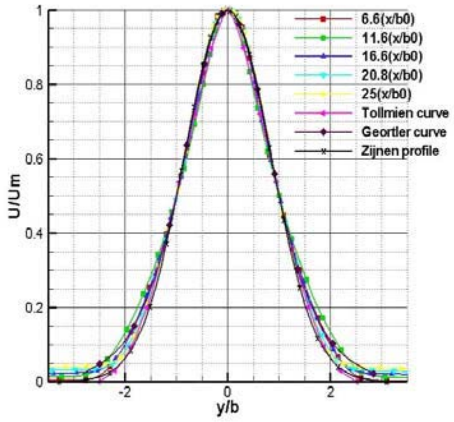
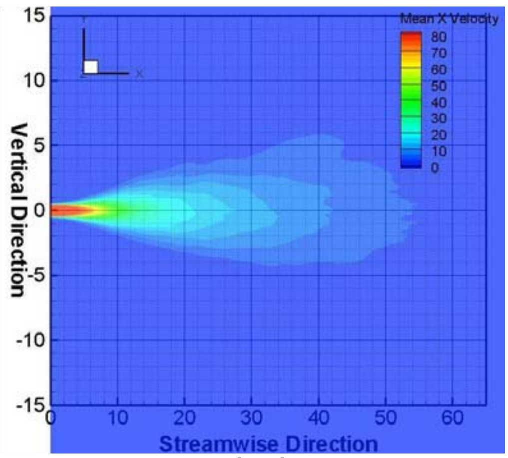


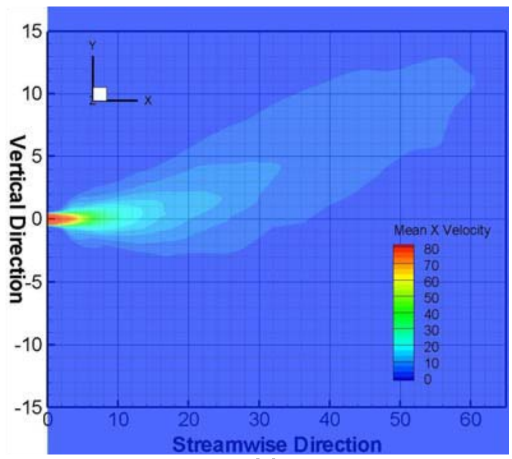
Figure 1. Vertical velocity comparison between CFD and theoretical methods for non-buoyant free jet

The spreading of the jet was found to be linear in the case of non-buoyant release and followed very closely the theoretical predictions. The buoyant jet, on the other hand, can be divided into three different linear regions:

- The jet region, closest to the exhaust, where the linear curve matches closely the non-buoyant line.
- The plume region, furthest away from the exhaust, where a high spreading of the jet can be observed because the buoyancy effect is dominant.
- The intermediate region, where the flow undergoes a transition from pure jet-like to plume-



Free jet without buoyancy



Free jet with buoyancy

Figure 2. Mean velocity profile comparison between buoyant and non-buoyant free jet

 ISBN: 978-0-615-20720-9

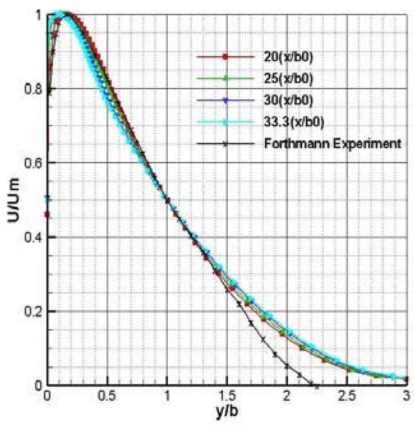


Figure 3. CFD and experimental results for the non buoyant wall jet vertical velocity profile

like behavior.

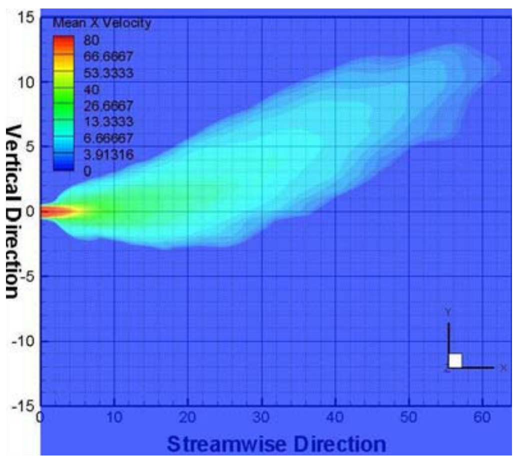
Vortices around the spanwise direction play an important role in the jet dispersion, as they regulate the potential core length when the counter-rotating vortices collide in the centerline axis and help the buoyancy jet to rise higher than the non-buoyant jet.

A second study within the ALAQS project aimed to provide an understanding of the impact of the presence of the ground on the fluid dynamics of the jet [21]. Before carrying out such comparison, a validation similar to the one presented in [20] was done on this wall jet against existing experimental and analytical results for a non-buoyant condition. Only then the buoyant case was analyzed and used to assess the impact of the solid boundary on the fluid mechanics of the jet flow. Once again, the results of the CFD simulation of the non-buoyant wall jet agreed very well with the classical results. An illustration of this is presented in Fig. 3, where it successfully replicated the boundary layer profile created by the wall and the free shear layer profile generated by the ambient fluid.

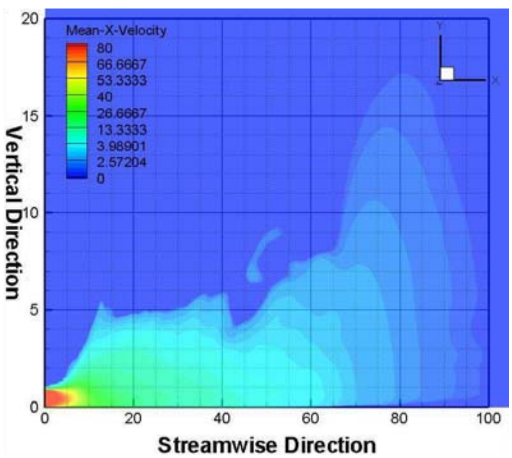
The comparison between the buoyant free and wall jets (Fig. 4) revealed several differences. First, the potential core was found to be much longer for the wall jet than for the free jet. This has an effect on the flow penetration through the control volume; the wall jet offers a deeper penetration than the free jet. The maximum velocity decays much faster for the free jet than for the wall jet. This leads to a correlation between this parameter and the penetration properties previously discussed, as the penetration involves higher velocity pushing into the control volume.

There is also an interconnection between all the parameters discussed previously and the streamwise vortical structure of the buoyant wall jet. As in the case of the buoyant free jet, counter-rotating vortices are created on one side by the surrounding fluid and on the other by the solid boundary. What is different from the free jet situation is the presence of the wall generating continuous vortices, whereas the influence of the vortices created by the surrounding fluid gradually decreases.

The first point of merging of the counter-rotating vortices occurs in the potential core. As the flow progresses, the intensity of the vortices generated by the wall is much stronger than the ones created by the surrounding fluid, causing a clinging of the flow, also known as the Coanda effect. This pushing-down phenomenon restricts the growth of the jet vertically, hence creating a lower rate of spread than for the buoyant free jet. As the velocity further away from the jet exhaust decreases, the vortices created by the wall decrease and buoyancy takes over, with positive streamwise vortices lifting up the flow from the ground.



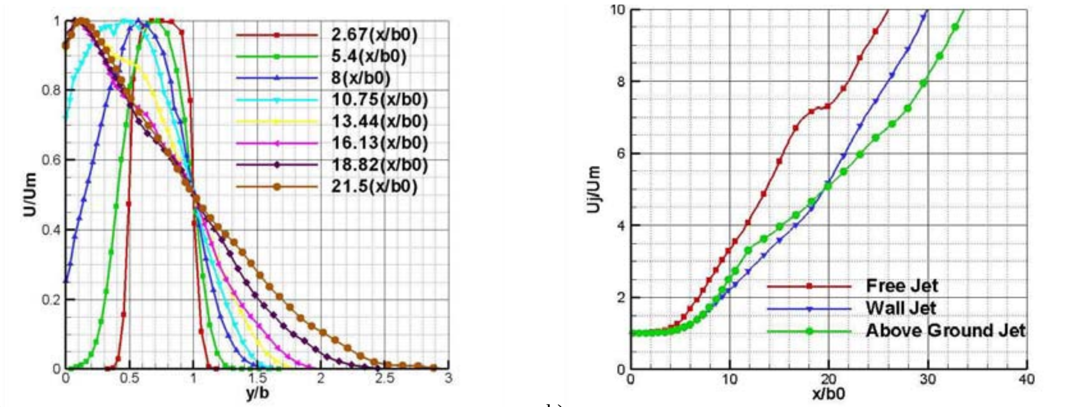
Free jet



Wall jet

Figure 4. Comparison of mean velocity profile of buoyant free and wall jet after 10s

ISBN: 978-0-615-20720-9



a) b)

Figure 5. Vertical velocity profile (a) and velocity decay comparison between different jets (b)

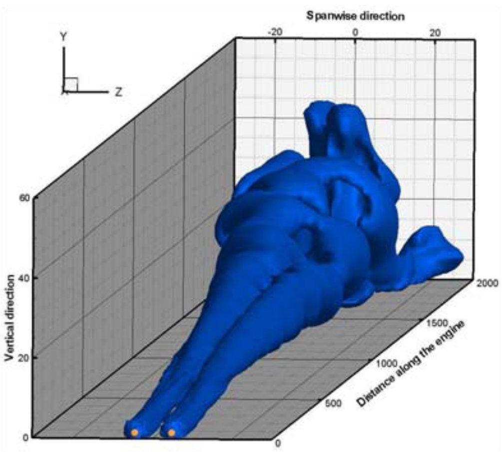
Raising the jet above the ground showed a combination of both free and wall jet velocity profiles. Near the engine exhaust, the velocity profile resembles that of the free jet whereas further downwind the characteristic self-similar profile of the wall jet can be found (Fig. 5a). The rate of decay of the maximum velocity of different jets is shown in Fig. 5b, with the above ground jet having the lowest rate. This is in line with the results reported by Davis and Winarto [22] and has a consequence in terms of penetration trough the control volume. The above ground jet configuration is expected to have a deeper penetration of the exhaust gases through the control volume.

A further study within the ALAQS project aimed at characterizing the effects of engine geometry and acceleration in the source dynamics by incorporating the complete geometry of an engine (CFM56-3C-1) [23]. Two engine configurations (single and double engine) were simulated under two types of wind (a headwind of 2.5m/s and a crosswind of 4.5m/s at an angle of 40 degrees from the engine centre). The results highlighted the importance of several parameters on plume dispersion.

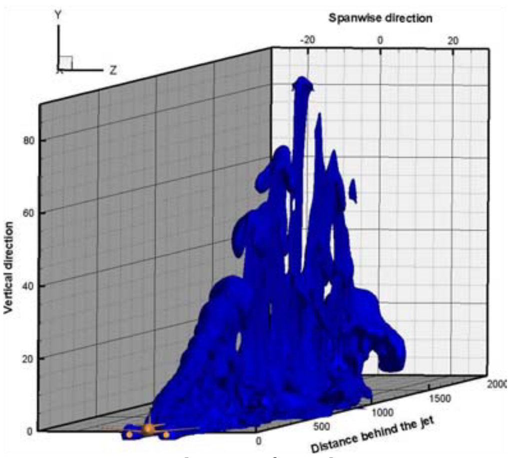
The first parameter is the wind configuration; headwinds directly applied to the engine will increase the plume penetration downwind. Crosswinds, on the other hand, increase the lateral spread of the plume. The second parameter is related to the vortical structure configuration. After the jet regime, divergence of the jet core from its centre occurs when the plume starts to weaken. This is the result of counter-rotating vortices wrapping around the "self-induced" rotation of the jet. A sinusoidal instability pattern was reported due to the divergence from the centre, which increases in amplitude and period as the flow progresses through the control volume, leading to the breakup of the instability.

In the case of a double engine, the wind configuration and vortical structures parameters still affect the plume dispersion process. In fact, it was found that the presence of two engines enhanced it because of the interaction between the two jets' plumes favoring a "straining" mechanism, resulting in the development of the instabilities (increase in amplitude), thus facilitating the breakup.

The final step was to include the full geometry of the airplane; a Boeing 737 was chosen because it uses the CFM56-3C-1 engines and is the most popular at medium size airports such as Zurich. Presently, results are only available with a double engine on the runway with a headwind condition. The results show some differences when the body, the wings and

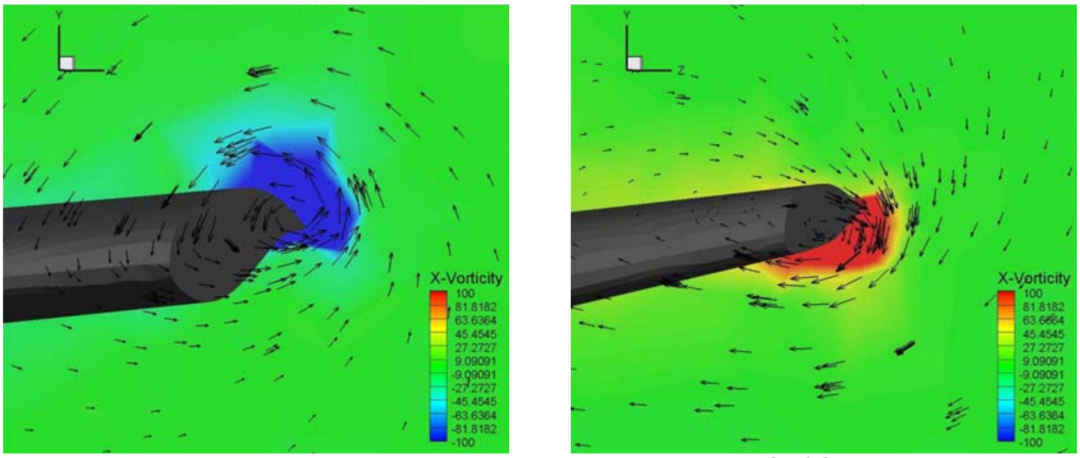


Double engine only simulation



Complete aircraft simulation

 ISBN: 978-0-615-20720-9



Wing tip vortices Horizontal stabilizer tip vortices

Figure 7. Wing and horizontal stabilizer tip vortices comparison

the empennage of an aircraft are added to the simulation.

Fig. 6 represents an iso-surface of 1 ppm NOx concentration. The whole aircraft restricts the flow progress downwind through the control volume. The penetration length is about 1,860m behind the exhaust, whereas for the engines alone it is extended by 40m. Without any geometrical disturbances, the flow was allowed to travel deeper.

On the other hand, some geometrical disturbances were found to enhance the vertical and lateral dispersion. These are due to the vortices created by the wing and the horizontal stabilizer. Fig. 7 shows a comparison of their tip vortices; the arrows represent the flow pattern in the Y-Z plane and the magnitude of the vortices is shown by the contour plot scale. The differences concern the rotation of the flow; the air stream over the wing creates a negative rotation at the tip around the X-direction, whereas the horizontal stabilizer's tip shows a positive rotation. This is expected as the horizontal stabilizer is in fact an inverted wing, but what is least expected is the impact they have on the plume dispersion.

Because the horizontal stabilizer is located just after the jet exhaust and almost on its way, it will be the first to affect the flow. Its disturbance is characterized by an enhancement of the vertical dispersion as can be seen in Fig 8. Part of the engine exhaust is attracted by the rotation of the fluid created by the horizontal stabilizer's tip and spreads upward.

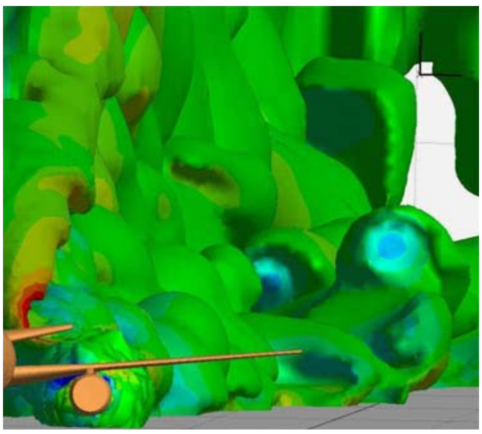


Figure 8. Iso-surface of 1ppm of NOx plotted with x-vorticity

The influence of the wing-tip can only be observed further away from the engines' exhausts when the jet plume has spread sufficiently laterally to meet the wing-tip vortices core. Its disturbance of the flow pattern is characterized by an enhancement of the lateral dispersion. Fig. 8 clearly shows this; the concentration of NOx rolls around the negative x-vortices core and spreads laterally, at low distances from the ground.

Other parameters of interest, such as velocity, temperature and turbulence, can be obtained from the CFD simulation. These are the major parameters to characterize the source dynamics in any problem. In addition, the simulation could serve several other purposes, e.g. related to air traffic operations. Results can shed light on the strength of the wake turbulence when different types of aircraft take-off and land. As a consequence, a proper time delay separation can be obtained for different classes of aircraft, thus increasing the airport traffic capacity.

The need for a better characterization of source dynamics in an airport environment is important not only for moving aircraft during the take-off and landing phases, but also necessary when they are immobile and using their APU. In this study, a non-reactive pollutant was introduced to simplify the problem but chemistry models exist for CFD simulations that allow the prediction of chemical transformations associated with plume dispersion.

VI. CONCLUSIONS

The work reported in the present paper is related to the need from airport dispersion modelers for better source dynamics characterization. Gaussian and Lagrangian techniques are well suited for complete airport dispersion calculations, but only include very simplified models to represent moving sources during take-off and landing. Unfortunately, these are not appropriate for adequate characterization of the near-field fluid dynamics. Conversely,

 ISBN: 978-0-615-20720-9

CFD techniques are presently not appropriate for the simulation of a full scale airport, but it can help to improve the understanding of some of the flow characteristics, including source dynamics.

This paper summarizes our CFD work related to airport local air quality studies. The strategy adopted was to use a staged approach, starting from a single engine free in the atmosphere and increasing the simulation complexity step by step. This allowed an initial validation with known theoretical and experimental results. After investigating free jets in buoyant and non-buoyant conditions, a wall jet simulation was analyzed and validated before the jet was raised to a proper distance above the ground. Important results of the simulations include the influence of the maximum velocity decay in the plume penetration through the control volume, and the role of counter-rotating vortices in the dispersion process.

The incorporation of the engine and the aircraft body, including the wings and empennage, revealed the effects of several other parameters such as the horizontal stabilizer's and wing's tip vortices. These were found to play an important role in the vertical and lateral dispersion, whereas the aircraft was found to slow down the plume penetration through the control volume.

This paper has demonstrated the applicability of CFD methods to airport-related flow and dispersion problems, and their capability to characterize source dynamics. The results can help dispersion modelers with better source dynamics representation and benefit the air traffic management of aircraft time separation delays in the take-off and landing phases.

ACKNOWLEDGMENT

The work reported in this paper has been undertaken as part of the Airport Local Air Quality Studies (ALAQS) project commissioned and sponsored by EUROCONTROL. The authors would like to thank all the members of the ALAQS consortium, especially Dr Ian Fuller from EUROCONTROL.

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] K. Schäfer, C. Jahn, P. Sturm, B. Lechner and M. Bacher. "Aircraft Emission Measurements by Remote Sensing Methodologies at Airports". Atmospheric Environment, Vol. 37, pp. 5261-5271. (2003).

[3] 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.

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

[5] P. Zannetti. "Air Pollution Modeling: Theories, Computational Methods and Available Software", Avon, UK, Computational Mechanics Publications, 1990.

[6] F. Farias and H. ApSimon. "Relative Contribution from Traffic and Aircraft NOx Emissions to Exposure in West London", Environmental. Modelling and. Software, Vol.21, pp. 477-485. Feb. 2005.

[7] E. Fleuti and P. Hofmann. "Airport Local Air Quality Studies: Case Study Zurich Airport 2004", Eurocontrol report. Eurocontrol Experimental Centre. (2005).

[8] E. Fleuti, P. Hofmann and C. Talerico. "Airport Local Air Quality Studies, Sensitivity Analysis Zurich Airport 2004", Eurocontrol report C21197/05. Eurocontrol Experimental Centre. (2006).

[9] A. Celikel, N. Duchene, I. Fuller, M. Silue, E. Fleuti, P. Hofmann and T. Moore. "Airport Local Air Quality Studies, Case Study: Emission Inventory for Zurich Airport with Different Methodologies. Eurocontrol report EEC/SEE/2004/010 . Eurocontrol Experimental Centre. (2004).

[10] D. Carruthers, C. McHugh, M. Jackson and K. Johnson. "Developments is ADMS-Airport to Take Acoount of Near Field Dispersion and Applications to Heathrow Airport". Proceedings of the 11^{th} International Conference on Harmonisation within Atmospheric Dispersion Modelling for Regulatory Purposes. (2007).

[11] U. Janicke and I. Fuller. "Derivation of Smooth and Shift Parameters to Account for Source Dynamics in ALAQS-AV Emissions Grids". Eurocontrol report C21197/05. Eurocontrol Experimental Centre. (2005).

[12] R.J. Yamartino, P. Builtjes and R. Stern. "Status of the Current Level of Development and Understanding in the Field of Modeling Pollutant Dispersion at Airports". UFOPLAN-Ref. No. 20341253/01. (2004).

[13] N. Duchene, I. Fuller and U. Janicke. "Verification of ALAQS Hourly 3D Grid Source Approach with Smooth and Shift Parameters to Account for Plume Dynamics". Proceedings of the 11^{th} International Conference on Harmonisation within Atmospheric Dispersion Modelling for Regulatory Purposes. (2007).

[14] K.N. Yu, Y.P. Cheung and R.C. Henry. "Identifying the Impact of Large Urban Airports on Local Air Quality by Non Parametric Regression". Atmospheric Environment, Vol. 38, pp. 4501-4507. (2004).

[15] R.L. Wayson, G.G. Fleming, B. Kim, W.L. Eberhard and W.A. Brewer. "Final Report: The Use of LIDAR to Characterize Aircraft Initial Plume Characteristics". FAA-AEE-04-01. (2004).

[16] W.L. Eberhard, W.A. Brewer and R.L. Wayson. "LIDAR Observation of Jet Engine Exhaust for Air Quality" 2^{nd} Symposium on LIDAR Atmospheric Applications. (2005). http://ams.confex.com/ams/pdfpapers/83405.pdf.

[17] Project for the Sustainable Development of Heathrow. Report for the Department for Transport. http://www.dft.gov.uk/pgr/aviation/environmentalissues/heathrow/. (2006).

[18] R.S. Sokhi, N. Kitwiroom and L. Luhana. "FUMAPEX-Datasets of Urban Air Pollution Models and Meteorological Pre-processors". Assessment of Different Existing Approaches to Forecast UAP Episodes". EVKL-CT-2002-00097. (2002).

[19] S. Aloysius, D. Pearce, L.C. Wrobel and M. Silue. "ALAQS-Comparison of CFD with Lagrangian based Simulations for Airfield Emissions Dispersion". Eurocontrol report. Eurocontrol Experimental Centre. (2006).

[20] S. Aloysius and L.C. Wrobel. "ALAQS- CFD Comparison of Buoyant and Non-Buoyant Turbulent Jets". Eurocontrol report. Eurocontrol Experimental Centre. (2007).

[21] S. Aloysius and L.C. Wrobel. "ALAQS- CFD Comparison of Buoyant Free and Wall Turbulent Jets". Eurocontrol report. Eurocontrol Experimental Centre. (2007).

[22] M.R. Davis and H. Winarto. "Jet Diffusion from a Circular Nozzle above a Solid Plane". Journal of Fluid Mechanics. Vol. 101, part 1, pp.201-222. (1980).

[23] S. Aloysius and L.C. Wrobel. "CFD Simulation of Emissions Dispersion from an Aircraft Engine during Take-off". Eurocontrol report. Eurocontrol Experimental Centre. (2006).

ISBN: 978-0-615-20720-9