ALAQS
Comparison of CFD and Lagrangian Dispersion Methods – Simple Scenario during Take-off

EEC/SEE/2007/001
ALAQS - Comparison of CFD and Lagrangian Dispersion Methods – Simple Scenario during Take-off

This report was prepared for EUROCONTROL Experimental Centre ALAQS by: School of Engineering and Design, Brunel University, UK.

Author(s): Syoninus Aloysius, Daniel Pearce, Porf. Luiz Wrobel

Ian Fuller, EUROCONTROL Experimental Centre


This document is published by EUROCONTROL in the interest of the exchange of information. It may be copied in whole or in part providing that the copyright notice and disclaimer are included. The information contained in this document may not be modified without prior written permission from EUROCONTROL.

EUROCONTROL makes no warranty, either implied or express, for the information contained in this document, neither does it assume any legal liability or responsibility for the accuracy, completeness or usefulness of this information.
EXECUTIVE SUMMARY

This report aims to simulate the dispersion of NOx from an aircraft engine during the take-off phase, and to provide information on source dynamics for future improvements of existing dispersion models. The dispersion of a single engine, typical of the B737, was simulated first, followed by the simulation of a twin engine. The results are discussed in detail and compared with results from other papers dealing with related problems, in order to correlate some of the findings.

The method employed to characterise the plume dynamics employs state-of-the-art Computational Fluid Dynamics (CFD) software packages, which represent the most advanced mathematics that can be applied to the simulation of fluid flow. The Large Eddy Simulation (LES) turbulence model was used to model the smaller eddies present within the grid resolution, and to solve explicitly the large eddies contained outside it.

The conclusions from the report highlight the importance of two parameters, namely wind data and vortical structures, for both simulations. Another important parameter for the double engine simulation relates to the interaction effects between the two engines.

Future studies will include a complete aircraft geometry. The implication of the results will also be of benefit to air traffic management, for the efficient prediction of time separation delays between aircraft in the take-off and landing phases.
This report presents the results of studies on emissions dispersion from an aircraft engine during the take-off phase. Two types of wind configuration were applied to a single and a double jet engine. Results highlight the influence of different parameters on the plume dispersion pattern, and confirm that CFD is an appropriate tool for source dynamics simulations.
(This page intentionally blank)
# TABLE OF CONTENTS

1 INTRODUCTION ................................................................................................................................. 1

2 CFD simulation of an engine during take-off ................................................................. 2
   2.1 Geometry and boundary conditions set up ................................................................. 2
   2.2 Analysis of plume dispersion after 1 second ............................................................. 6
      2.2.1 Concentration of NOx .................................................................................... 6
      2.2.2 Vortical structures study ............................................................................. 8
   2.3 Analysis of plume dispersion after 10 seconds .......................................................... 10
      2.3.1 Concentration of NOx .................................................................................. 10
      2.3.2 Vortical structures study ............................................................................. 12
   2.4 Analysis of plume dispersion after 42 seconds .......................................................... 15
      2.4.1 Concentration of NOx .................................................................................. 15

3 CFD Simulation of two engines during the take-off phase ................................................ 17
   3.1 Analysis of plume dispersion after 1 second ............................................................. 17
      3.1.1 Concentration of NOx .................................................................................. 17
      3.1.2 Vortical structures study ............................................................................. 17
   3.2 Analysis of plume dispersion after 10 seconds .......................................................... 18
      3.2.1 Concentration of NOx .................................................................................. 18
      3.2.2 Vortical structures study ............................................................................. 18
   3.3 Analysis of plume dispersion after 42 seconds .......................................................... 19
      3.3.1 Concentration of NOx .................................................................................. 19

4 CONCLUSIONS ................................................................................................................................. 19

5 REFERENCES ................................................................................................................................. 21

APPENDIX A ......................................................................................................................................... 23

APPENDIX B ......................................................................................................................................... 24

APPENDIX C ......................................................................................................................................... 26

APPENDIX D ......................................................................................................................................... 28

APPENDIX E ......................................................................................................................................... 31
LIST OF FIGURES

FIGURE 1 CHEMICAL COMPOSITION OF A JET EXHAUST AND PERCENTAGES ASSOCIATED WITH EACH COMPONENT [3] ................................................................. 1
FIGURE 2 CFD GEOMETRY AND MESHING FOR SINGLE AND DOUBLE ENGINE SIMULATIONS .... 3
FIGURE 3 HORIZONTAL DISPERSION FOR SINGLE ENGINE SIMULATION AFTER 1 SECOND ......... 6
FIGURE 4 VERTICAL DISPERSION FOR SINGLE ENGINE SIMULATION AFTER 1 SECOND ............ 8
FIGURE 5 CONCENTRATION ISO-SURFACE OF 15PPM PLOTTED WITH X-VORTICITY AFTER 1 SECOND FOR SINGLE ENGINE SIMULATION ........................................ 9
FIGURE 6 ADDITION OF ISO-SURFACE OF -10/S AND 10/S X-VORTICITY ......................................... 10
FIGURE 7 HORIZONTAL DISPERSION FOR SINGLE ENGINE SIMULATION AFTER 10 SECOND ...... 10
FIGURE 8 VERTICAL DISPERSION FOR SINGLE ENGINE SIMULATION AFTER 10 SECONDS ........ 11
FIGURE 9 ISO-SURFACE OF 12PPM OF NOX CONCENTRATION PLOTTED WITH X-VORTICITY .... 12
FIGURE 10 ISO SURFACE OF 10.5PPM PLOTTED WITH -3/S AND 3/S X VORTICITY ..................... 13
FIGURE 11 ISO-SURFACE OF 10.5PPM AND -3/S AND 3/S X VORTICITY ........................................... 13
FIGURE 12 CROW VORTICES FORMATION [18] ............................................................................. 14
FIGURE 13 HORIZONTAL DISPERSION FOR SINGLE ENGINE SIMULATION AFTER 42 SECONDS ... 15
FIGURE 14 VERTICAL DISPERSION FOR SINGLE ENGINE SIMULATION AFTER 42 SECONDS ....... 16
FIGURE 15 ISO-SURFACE OF CONCENTRATION PLOTTED WITH X-VORTICITY AFTER 1 SECOND FOR DOUBLE ENGINE SIMULATION .............................................. 17
FIGURE 16 ADDITION OF ISO-SURFACE OF -10/S AND 10/S X-VORTICITY FOR DOUBLE ENGINE HEADWIND SIMULATION ......................................................... 18
FIGURE 17 ISO-SURFACE OF 12PPM OF NOX CONCENTRATION PLOTTED WITH X-VORTICITY FOR DOUBLE ENGINE ........................................................................ 19

LIST OF GRAPHS

GRAPH 1 HORIZONTAL DISPERSION FOR SINGLE ENGINE SIMULATION AFTER 1 SECOND .......... 7
GRAPH 2 VERTICAL DISPERSION FOR SINGLE ENGINE SIMULATION AFTER 1 SECOND ............... 8
GRAPH 3 VORTICITY DISTRIBUTIONS AT 56.5M BEHIND THE ENGINE ........................................ 9
GRAPH 4 HORIZONTAL DISPERSION FOR SINGLE ENGINE SIMULATION AFTER 10 SECONDS ...... 11
GRAPH 5 VERTICAL DISPERSION FOR SINGLE ENGINE SIMULATION AFTER 10 SECONDS .......... 12
GRAPH 6 HORIZONTAL VORTICITY DISTRIBUTIONS AT 286.5M BEHIND THE ENGINE FOR THE HEADWIND SIMULATION ......................................................... 14
GRAPH 7 VERTICAL VORTICITY DISTRIBUTIONS AT 286.5M BEHIND THE ENGINE FOR THE CROSSWIND SIMULATION ......................................................... 15
GRAPH 8 HORIZONTAL DISPERSION FOR SINGLE ENGINE SIMULATION AFTER 42 SECONDS ...... 16
GRAPH 9 VERTICAL DISPERSION FOR SINGLE ENGINE SIMULATION AFTER 42 SECONDS .......... 17
1 INTRODUCTION

Pollution resulting from airport emissions is a growing concern because of the expansion of air traffic over the years. Future air traffic movements are forecast to grow at a mean annual rate of 5 to 7 percent [1].

Even though most engines have become more fuel efficient and less pollutant, the expansion of air traffic more than compensate for this reduction, maintaining the high levels of emissions locally and globally. Local problems created by high levels of emissions are poor air quality and environmental phenomena like smog [2].

Local pollution also contributes to global problems such as the depletion of the ozone layer and global warming. According to the international panel on climate change, air transportation accounts for 3.5 percent of global warming from human activity. The same report predicts a rise to 15 percent if no restrictions are imposed by 2050 [3].

![Chemical composition of a jet exhaust and percentages associated with each component](image)

**Figure 1 Chemical composition of a jet exhaust and percentages associated with each component** [3]

Thrust for a jet aircraft is provided by its engines through hot gases pushed out off the exit nozzle. Combustion is a necessary process for the engine to work. In an ideal situation, the combustion of aircraft fuel will only produce carbon dioxide (CO\(_2\)) and water (H\(_2\)O) [3], as shown in Figure 1. In real conditions, residual products such as soot, hydrocarbons (HC) and carbon monoxides (CO) will form from “non-ideal combustion processes” and from the oxidation of nitrogen contained in air, the so-called nitrogen oxides (NO\(_x\)) [4].

Nitrogen oxides represent the largest portion of unwanted chemical reactions and are mostly influenced by the ambient inlet conditions and the power setting. As a matter of fact, NO\(_x\) emissions are mostly generated when the engine is running at a very high temperature (i.e. during take off) [3]. Dispersion resulting from these processes is often not properly incorporated in the most commonly used monitoring software, because these processes are in a time scale of only a few minutes [5].
The software LASPORT incorporates, at present, a smooth and shift approach to account for aircraft source dynamics, which is based on LIDAR measurements carried out at Dusseldorf airport. At the moment, the same parameters are used for all types of aircraft [5].

Computational Fluid Dynamics (CFD) is considered to be the most appropriate technique to use in near-field source dynamics and interactions in airport-related problems [6]. This report investigates the dispersion process of an aircraft engine during the take-off mode using an advanced CFD tool.

A control volume including the runway length of a particular airplane and a dispersion length was used. The terrain is assumed to be flat, and only the engine is modelled emitting passive NOx at its exhaust. The take-off mode was simulated using an acceleration value at the inlet of the control volume.

This report will first present the results for a single aircraft engine (representative of a Boeing 737) in a take-off configuration, for both a headwind and a crosswind. The second section will use the same setup but with two engines modelled on the runway.

2 CFD SIMULATION OF AN ENGINE DURING TAKE-OFF

This report is part of the Airport Local Air Quality Studies (ALAQS) project. The purpose of this project is to develop information designed to assist airport regulators and operators in improving local air quality [7]. The aim of this particular report is to present the results obtained from CFD simulations to improve the source dynamics representation in models based on Lagrangian techniques.

2.1 Geometry and boundary conditions set up

The computational domain includes the runway length (1,600m taken from the B737 handbook chart [8] for a standard day at the Zurich airport elevation of 1,416 ft and a maximum take-off weight of 63,000 kg), an engine corresponding to the one most commonly used in the B737 (CFM56-3C-1) and a downwind area for dispersion purposes. The whole domain is split into two major volumes (four for the two engine simulation) of different sizes:

- The engine volume: This volume contains four (eight for the two engine simulation) sub-volumes with three structured and one unstructured mesh. The unstructured sub-volume describes the overall geometry of the Boeing 737 and contains the engine at the break release point. This main volume has a fine mesh density, about 150,000 cells (250,000 cells for two engine simulation), as initial dispersion will take place around this area after several milliseconds.

- The downwind volume: This volume is also divided into four (eight for the two engine simulation) sub-volumes, all containing structured meshes. It contains the remaining runway length and a further dispersion length to account for the plume dynamics. Since the dispersion process is the main interest of this report, a very fine mesh density was assigned to those volumes (about 760,000 cells for the single engine, 1,410,000 cells for the double engine) as required by Large Eddy Simulation models and discussed in a previous report [6].
The only paper found in the literature that has attempted to simulate airplane pollutant dispersion during take-off using CFD is a recent one by Koutsourakis et al. [9]. Their control volume for the simulation of a Boeing 737-400 engine is $3,500 \times 500 \times 150$ in the $x$, $y$, $z$ directions, respectively. The mesh adopted had uniform spacing with 70 nodes in the $x$-direction, non-uniform spacing (finer near the runway) with 44 nodes in the $y$-direction, and non-uniform spacing (finer closer to the ground) with 31 nodes in the $z$-direction. The total mesh comprises nearly 96,000 cells. A 3D unsteady RANS simulation was carried out using in-house software called ADREA-HF.

LES turbulence models require an appropriate mesh density in order to capture eddies of acceptable size. A compromise is necessary because a finer mesh would require a reduction in time step and an increase in computational time, while a coarser mesh would reduce the computational time but some of the fluid mechanics characteristics would be omitted. Two-dimensional tests were carried out, with both horizontal and vertical cross-sections, to determine a suitable mesh distribution. It was found that a mesh with 910,000 cells (for the single engine) and 1,660,000 cells (for the double engine) was adequate for both simulations. These tests are described below.

Each study involved four cases using different mesh numbers and spacing ratio. The tables below show the mesh setup selected for the study, producing both fast and accurate simulations.
The strategies adopted for this comparison study were as follows:

1. The engine moves with acceleration through the whole computational domain via a dynamic mesh approach. This allows, by proper coding, the simulation of the engine movement along the runway, but requires the remeshing of the computational domain around it with a specified length scale and cell skewness criteria.

2. The engine is stationary on the runway. The inlet velocity changes via a user-defined function to replicate the engine acceleration, as described above.

Unfortunately, the first test could only be conducted for a period of 8 seconds because the dynamic meshing strategy is very computationally intensive.

The results from this study show that there is not much difference on the dispersion pattern and the level of pollutant magnitude at a short distance behind the engine. Some disturbances of very low magnitude occur for the downwind dispersion, mainly due to the vorticity patterns, as the second strategy shows a more elongated dispersion pattern.

Because the control volume is not closed, particular attention should be taken when selecting the boundary conditions.

The ground is setup as a wall with no slip condition, allowing a boundary layer to be formed. The side and top boundaries are assigned as symmetry boundaries. The engine boundary conditions were more difficult to set up, as the amount of data available does not correspond to the boundary conditions input available for the FLUENT software.

The engine casing is setup as a wall with an assigned temperature. The intake fan does not suck air into the engine, but allows the air just upwind to pass through, creating no perturbation of the flow. An intake temperature is set up using the data available in [4].

The mass flow inlet best represents the exhaust from the gas turbine. From available references [4] and [12], data were taken as mass flow rate 322kg/s, exhaust temperature 690K.

The species considered for the report is NOx, generated when the engine is running at very high temperature. Data collected from the International Civil Aviation Organization (ICAO) gives NOx emission for the CFM56-3C-1 as 20.7g/kg of fuel; the fuel flow is 1.154kg/s and the time in mode for take-off is 42s. Multiplying these values gives 1.003kg of NOx emitted during the take-off phase. Knowing the mass flow rate of the engine, the total mass of gas emitted can be calculated as $322 \times 42 = 13524$kg and by dividing the NOx emissions by the total emissions, the mass fraction of NOx is obtained as $0.74 \times 10^{-4}$.

In order to investigate the influence of meteorological conditions on the dispersion pattern behind the engine, Zurich airport was taken as the test scenario. The meteorological data kindly provided by UNIQUE Flughafen Zurich AG contain information on the wind direction and magnitude throughout a whole year. The study was conducted by selecting and computing results for a month for each season (January, April, July, and October). An average wind direction and speed was calculated considering every single day of the month.
In winter, major winds with high magnitude (highest being 7.4 m/s) come from the Northeast. Northwest winds are never higher than 4m/s. Some South-West winds with a direction of 73 degrees from the North have recorded a magnitude of 4.7m/s.

In spring, the wind magnitudes are lower than 5m/s. Winds are mostly concentrated in the Northern part, with higher frequency of occurrence in the Northwest part (direction of 100 degrees from the North with 4.6m/s). Only one very high magnitude wind blowing at 4.4m/s was observed from the Southwest.

In summer, all the wind vectors are concentrated in the Northeast with the highest magnitude reaching 3.5m/s. The highest wind for that month was a Northwest at about 160 degrees from the North.
In autumn, the winds are low in magnitude with nearly all of the vectors blowing from the Northeast. The highest wind for that particular month was about 7.1 m/s. A representative month of this season is almost similar to January, therefore only one of these scenarios will be simulated.

There are three main runways at Zurich airport; Boeings 737 can use any of them, depending on their take-off weight, the meteorological conditions and the airport elevation. A study of aircraft movements around the airport reveals that runway 16-34 is the most frequently used with the meteorological conditions discussed above.

As a consequence, two wind scenarios were taken for the simulation in accordance with the previous discussion:

- A headwind of 2.5 m/s
- A crosswind of 40 degrees from the engine centre with a magnitude of 4.5 m/s

The following section compares and evaluates the results of the CFD simulations for the particular time period. The scale used in all the figures representing the NOx concentration will remain unchanged throughout the report unless stated. Its magnitude varies from a zero value in blue to a maximum value of 1 ppm, represented in red for more clarity.

2.2 Analysis of plume dispersion after 1 second

2.2.1 Concentration of NOx

Horizontal dispersion:

The two figures above represent contour plots of NOx. It can be seen there are no major differences in terms of dispersion pattern and magnitude between the headwind and crosswind simulations, although the headwind plume has a longer spread. The velocity of the headwind simulation is slightly higher in the streamwise direction than the crosswind simulation. This has a
small effect in the dispersion length, with higher winds in the streamwise direction helping the
downwind diffusion, hence producing longer plume length. This conclusion is supported by the
studies of exhaust flow dispersion from a motor vehicle by Ning et al. [13], who observed a
deeper penetration of the jet downstream for higher exit velocity flows. In addition, Chan et al.
[14] and Wang et al. [15] showed that plume dispersion along the streamwise direction is highly
dependent on the “wind direction and magnitudes of velocity”.

The following graph of pollutant concentration at different locations behind the engine also
corroborates the above findings, clearly showing that after a distance of 56 meters behind the
engine, there is a lower magnitude of pollutants for the crosswind simulation whereas the
headwind simulation shows higher and more centered dispersion.

The graph also shows slight differences occurring at about 46 meters behind the engine, where
the crosswind simulation shows a deviation of the flow from the centreline of the engine. This
change in direction is obviously due to the crosswind velocity, but it can be noted that this
process happens far downwind (approximately 56 meters behind the engine).

Although Laatar et al. [16] discussed the importance of the wind direction in finding both the
concentration distribution and the topology of the flow in their 2D LES simulation of pollutant
dispersion around a covered roadway, it was actually Wang et al. [15] who demonstrated that,
along with exhaust plume development, the effect of wind velocity plays a significant role in the
dispersion process.

![Graph 1: Horizontal dispersion for single engine simulation after 1 second](image)

**Vertical dispersion:**

Similar observations are also valid for vertical plume dispersion. The NOx contour plots along the
engine centre line in Figure 4 show that the headwind and crosswind simulations predict the
same magnitude and dispersion patterns, with a small difference in dispersion length. Again, the
headwind simulation predicts a longer plume spread downwind than the crosswind simulation,
which is attributed to wind effects.
The graph below shows that near the engine the dispersion pattern and magnitude are exactly the same, the differences occurring at approximately 46 meters behind the engine where the crosswind simulation shows a more buoyant plume with higher concentration at higher altitude than the headwind simulation.

This point was also discussed by Ning et al. [13] who stated that, for a lower streamwise velocity, the jet momentum will die early and by doing so will facilitate the vertical dispersion of the plume. The headwind results still show vertical dispersion but this process happens further downwind.

Chan et al. [14] highlighted the importance of the ‘vortex circulation effect’ on pollutant dispersion. They concluded that the variations of pollution concentration distribution in a control volume are principally due to the circulation of vortices.

The following section shows some evidence that vortical effects have a significant influence on the plume dynamics, but this fact will become more apparent as the plume progresses with time.

2.2.2 Vortical structures study

Very few studies have been found in the literature related to vortex dynamics near the ground. Most studies concern the wake-vortex created by an airplane at high altitude, related to problems of contrail developments.

Spalart [17] reviewed current work on airplane trailing vortices and highlighted the fact that vortex behaviour near the ground is only understood qualitatively, and further research needs to be
done to help Air Traffic Controllers (ATC) to shorten or delay the separation between aircrafts during the landing and take-off phases (LTO).

Vortices are a result of turbulence generated by the fluid when encountering some disruption of its flow pattern, for instance a rough wall. As an example, wing tip vortices are formed by the reaction force (lift) generated by a pressure difference created on the wing. For an engine jet, the same mechanism applies and Jacquin [18] found that, in fact, there is a contra-rotating vortex pair that forms and conveys the momentum of the jet.

Even though taken at a very early time, Graph 3 below shows some correlation between vortices and emissions concentration in the sense that both have similar lengths. Another interesting points to note are the double peak for the vorticity magnitude and the point where there is a change in the sign of the vorticity. They actually correspond to a position near the centre line of the engine. The fluctuations can be associated with the findings of Jacquin [18], corresponding to the contra-rotating vortex pair discussed above.

Figure 5 shows the effects of the x-vorticity on emissions dispersion. The figures represent an iso-surface of 15ppm of NOx, plotted with vorticity around the x-axis. They both show the same dispersion pattern, and that some instabilities have been created at the tail of the plume. To understand this phenomenon more clearly, Figure 6 shows additional iso-surfaces of -10/s and 10/s x-vorticity, where it can be seen that the vortices around the x-axis wrap around the concentration iso-surfaces, thus causing a slight change of plume direction.
2.3 Analysis of plume dispersion after 10 seconds

2.3.1 Concentration of NOx

Horizontal dispersion:

From the engine exhaust to approximately 190 meters behind it, both contour plots in Figure 7 and the graph below show the same pattern and magnitude for the dispersion process. The differences start to appear at about 190 meters behind the engine, with the crosswind simulation displaying more lateral spreading, as expected, while the headwind simulation shows a longer dispersion pattern.

Graph 4 below shows some more differences behind the engine exhaust after 190m between the headwind and crosswind simulations, confirming the findings of Chan et al. [14] and Wang et al. [15] concerning the influence of wind magnitude and direction on the plume dispersion. The graph shows that these influences gradually increase along the streamwise direction.
Vertical dispersion:

From the inspection of both the above contour plots and Graph 5, the same observation can be drawn as for the horizontal dispersion, but here the similarities between both simulations end at about 190 meters behind the engine exhaust. The dispersion pattern and magnitude differ from that point onwards, with the crosswind simulation showing a much more buoyant plume rise than the headwind simulation. This is again related to the fact that the crosswind plume will not penetrate as much as the headwind simulation, enabling buoyancy effects to start much earlier.

The plume can be observed to rise to a height of 40 meters at a distance of 290 meters behind the engine, and to be in the order of several ppm. Similar height and concentration are achieved by the headwind simulation at about 360 meters behind the engine. At that distance, it can be seen that the concentration for the crosswind simulation is almost zero.

Buoyancy is an important factor in the dispersion process of the engine plume. Garnier et al. [19] showed a correlation between the buoyancy force and the dilution ratio through the mean temperature, and found that this happens after a distance of about "two or three wing spans". Gerz and Ehret [20] also confirmed in their analysis of wing tip vortices and jets exhaust interactions during the jet regime that it is the buoyancy force of the hot exhaust which controls vertical entrainment.
Besides agreeing with the influence of meteorological conditions on the dispersion process, Sini et al. [21] found that the so-called “gravitational thermal effects” also have an impact on the spread of emissions. In their work on street canyons, they concluded that both the dilution and penetration processes are dominated by a combination of two mechanisms, namely the turbulent diffusion and the temperature advection transport. This point will be further discussed later on in this report, but with the addition of other parameters specific to this case study. Sini et al. [21] also point out that the vertical transport process strongly depends on vortical structures.

The following section aims to find out how these effects influence the vertical transport.

2.3.2 Vortical structures study

After 1 second, some disturbances were created at the tail of the plume and it was stated that these were caused by the x-component of vortices wrapping around the concentration and thus causing the plume to slightly change its direction.
Figure 9 shows similar patterns and magnitudes of vorticity along the x-axis acting on the NOx concentration. Because of this, the following discussion will be applicable to both cases and thus only one simulation, for the headwind, can be taken to analyse the patterns obtained.

![Figure 10 Iso surface of 10.5ppm plotted with -3/s and 3/s x vorticity](image)

![Figure 11 Iso-surface of 10.5ppm and -3/s and 3/s x vorticity](image)

Figure 10 and 11 are taken from the headwind simulation; the scales were reduced in order to highlight some of the points that will be discussed next.

The first remark that can be made with regard to Figure 11 is the increase in the instability length compared to that of Figure 5 at 1 second. Figure 11 confirms the previous findings that the vortices around the x-axis are wrapping around the plume causing the instabilities, which have a sinusoidal pattern. These can be related to several papers on wake-vortex interaction, and some analogy can be drawn to explain these phenomena.

In this particular case, Lewellen et al. [22] observed that the engine exhaust is wrapped into the wing vortex forming a sinusoidal instability called Crow instability. Garnier and Laverdant [23] added that the break-up of this sinusoidal instability is due to the pair (because of the existence of two jets and two wings in an aircraft) mutual induction, also known as long wave instability. It must be mentioned here that their study was conducted at very high altitudes, meaning no interaction with the ground.

The ground plays an important role because it restricts the vertical flow. The analysis of Proctor and Switzer [24] speculates that there will be an upward rebound and divergence of the vortex pair in the presence of the ground.

Location A represented in Figure 10 indicates a breakdown of high concentration from the plume, due to instability. To understand such process, Figure 12 below taken from Jacquin [18] presents a sinusoidal instability and explains its evolution by a superposition of three mechanisms. Unfortunately, not all are applicable to a single engine, and only one of these effects can be considered for this study, the ‘self-induced rotation’ of a single sinusoidal instability [18].

The ‘self-induced rotation’ in Figure 12 creates contra-rotating vortices, one positive and one negative, making the core of the jet diverge from its centre. These, in turn, move in the z-direction apart from each other, making the instability increase in amplitude and period. Such growth will not last too long; at some point, there will be a break-up occurring close to the middle of the divergence, as represented in Figure 12. When this happens, one vortex will go toward the positive z-direction with the other going in the negative z-direction. This means that the dispersion
in the horizontal axis is due to the x-vortices. To prove such statement, a graph similar to Graph 3 above can be plotted.

Figure 7 showed that there are fluctuations of higher concentrations near the end of the plume of the headwind simulation. The fourth line from the left in the same figure displays the previously discussed characteristics, its position being at 286.5 meters behind the engine.

Graph 6 shows a vorticity distribution plotted at 286.5 meters behind the engine in the horizontal direction. It can be seen from this graph that there is a peak of concentration in the spanwise direction, similar to that shown in Figure 7. Moreover, the vorticity parameters reveal that it is in fact the vortices around the x-axis that are dominant over the other directions. This proves that the concentrations dispersed from the instability are carried away by the vortices around the x-axis.

In the same manner, it is possible to analyse the influence of parameters in the upward motion of the emissions plume. Even though the headwind shows concentration at higher altitudes, there is no fluctuation that can enable some correlation to be made. The crosswind simulation, on the
other hand, shows some high concentrations fluctuation, as presented in Graph 5, at about 286.5m behind the engine.

Graph 7 shows that there is a correlation between the vorticity pattern and NOx concentration. The vortices around the y and z-axis are dominant where there is a region of high concentration, whereas the magnitude of the vortices around the x-axis is very small in general compared to the other directions, except near the ground.

2.4 Analysis of plume dispersion after 42 seconds

Although the results for times between 10 to 42 seconds are not shown in this report, the findings at this particular time are representative and applicable throughout the simulation.

2.4.1 Concentration of NOx

Horizontal dispersion:

It is difficult to capture the whole dispersion length in one single figure, since it extends to approximately 2,040 meters. Because of this, the results in the above figure were zoomed to show the downwind area where the most interesting results can be found.
The statements made for earlier times are still valid, as can be seen in Figure 13. The headwind simulation presents a wider spreading in the horizontal direction because the wind velocity favours plume penetration, whereas the plume in the crosswind simulation stops well before but has a higher concentration in the lateral direction.

The influence of the wind magnitude and direction gradually increases along the streamwise direction, when the plume weakens. This point is reinforced in Graph 8, where both simulations display higher concentration in the first 700 meters but, further downwind, the influence of instabilities take place allowing lateral dispersion to occur.

Graph 8 Horizontal dispersion for single engine simulation after 42 seconds

Vertical Dispersion:

Figure 14 Vertical dispersion for single engine simulation after 42 seconds

Figure 14 confirms that the headwind simulation predicts a deeper emissions penetration because of the favourable wind velocity. The upward motion of the plume happens much earlier for the crosswind simulation, again due to the fact that the wind velocity is much lower in the streamwise direction than for the headwind simulation. Instabilities can then take place much easier and earlier for the crosswind simulation than for the headwind simulation. These can be correlated with Graph 9 below.
3 CFD SIMULATION OF TWO ENGINES DURING THE TAKE-OFF PHASE

3.1 Analysis of plume dispersion after 1 second

3.1.1 Concentration of NOx

Unfortunately, the findings in the single engine analysis are not so obvious for the horizontal and vertical dispersion of two engines after 1 second. As a matter of fact, both figures in APPENDIX B show that the dispersion pattern and magnitude for each engine are very similar to that produced by the single engine. However, it appears that the single engine jet offers a deeper plume penetration than the double engine. This point will be discussed in greater detail in the following section.

3.1.2 Vortical structures study

The correlation between vortices and pollutant concentration, and their basic interactions, will not be repeated here as this was previously done for the single engine analysis. This section will then focus on how the vortices behave in a double engine configuration. Because the same parameters were used as the single engine simulation (Figure 5), a direct comparison can be made in order to understand why the plume penetration is shorter in the double engine configuration.
It can be seen from Figure 15 that the instabilities, also found in the single engine (Figure 5), happen much earlier enabling them to slow down the concentration and penetrate further downwind, hence causing a shorter dispersion in the streamwise direction. These instabilities are again associated with the vortices around the x-axis, as shown in Figure 16.

![Figure 16 Addition of iso-surface of -10/s and 10/s x-vorticity for double engine headwind simulation](image)

3.2 Analysis of plume dispersion after 10 seconds

3.2.1 Concentration of NOx

From the engine exhaust to approximately 90 meters behind it, both figures in APPENDIX C show the same pattern and magnitude for the dispersion process. The differences start to appear at about 190 meters behind the engine, with the crosswind simulation spreading more laterally, while the headwind simulation has a longer dispersion plume further downwind.

A break up of concentration in the crosswind simulation can also be observed, enabling the plume to spread laterally, whereas for the headwind simulation the plume is still attached. As shown earlier in the report, this is caused by the break up of vortices around the x-axis, transporting the concentration laterally.

Similar to the single engine case, the main findings of the double engine analysis are as follows:

- There is a deeper plume penetration downwind when the wind is in the same direction of the jet flow;
- The crosswind simulation predicts more lateral spread in the wind direction;
- The influence of wind magnitude and direction on the plume gradually increases along the streamwise direction.

3.2.2 Vortical structures study

The objective of this study is to understand the implication of having two engines on the vortical structure pattern. It was demonstrated earlier on the report that the dispersion undergoes a sinusoidal instability created by the vortices around the x-axis. As these evolve, break up occur leading to lateral (concentrations transported by the vortices rotating around the x-axis) and vertical (concentrations transported by the vortices around y and z-axis) dispersion.
Comparing Figure 17 and Figure 10, it becomes apparent that there is an increase in amplitude from the start of the instability. The two-engine simulation offers an amplified sinusoidal fluctuation compared to the single engine simulation.

One of the mechanisms predicted by Jacquin [18], related to the evolution of the sinusoidal instability, was already found in the single engine simulation. Here, it can also be observed that ‘straining’ by each plume increase the perturbation, leading to the increase in amplitude.

Another previously mentioned feature is the onset of instability, clearly noted in Figure 17. Stability starts earlier for the crosswind simulation than for the headwind simulation. This, in turn, has an effect on the plume penetration length. It appears that a crosswind will trigger an early instability causing the flow to slow down the plume, hence generating shallow penetration further downwind.

3.3 Analysis of plume dispersion after 42 seconds

3.3.1 Concentration of NOx

The results obtained for the dispersion pattern after 42 seconds are presented in APPENDIX D. From these figures, it is clear that the headwind simulation predicts higher concentrations further downwind than the crosswind simulation.

In addition, it is possible to see that the lift-off of concentration from the ground occurs also for the two-engine simulation. It was previously stated that this was caused by the interaction between the vortices and the ground, causing the plume to rise. This lift-off is broken earlier for the crosswind simulation, resulting in an increase in concentration on the ground much earlier than for the headwind simulation. This supports the finding that a crosswind favours the break up of instabilities much quicker than a headwind, causing a shallower penetration and a larger spread of concentration downwind.

4 CONCLUSIONS

This report presented some results on emissions dispersion behind a single and a double engine. This information aims to improve the understanding of the flow behaviour for an aircraft in the take-off phase, and intends to provide an initial assessment on the use of CFD techniques as a tool to improve source dynamics representation.

The results highlighted the importance of several parameters that must be properly taken into account because of their major effects on plume dispersion, as follows:

- Wind configuration

Head winds directly applied to the engine will increase the plume penetration downwind. Crosswinds, on the other hand, increase the plume lateral spread. The initial source dynamics is governed by the plume straight jet; it is only further down, along the streamwise direction, that the influences of the wind configuration gradually increase.
Vortical structures

After the jet regime, when the plume starts to weaken, divergence of the jet core from its centre will occur. This is the result of contra-rotating vortices wrapping around the ‘self-induced rotation of the jet’. This divergence leads to instability of a sinusoidal pattern, which increase in amplitude and period as it progresses through the control volume. This increase will lead to break up of the instability. When this occurs, it was found that the vortices around the x-axis are the ones transporting the concentration laterally whereas the ones around y and z-axis help the vertical dispersion.

An additional parameter for the two-engine simulation is the influence of the interactions between the two engines. The instabilities are still apparent in the case of two engines, but in this case they form a pair. As the plume weakens, the interaction between the engines plumes favours the ‘straining’ mechanism, resulting in the development of the instabilities (increase in amplitude) and thus facilitating break-up.
5 REFERENCES


APPENDIX A

Iso-surfaces of NOx concentration after 42 seconds of dispersion behind a single engine

Headwind simulation

Crosswind simulation
APPENDIX B

Double engine analysis of plume dispersion after 1 second

Horizontal dispersion:

Headwind simulation

Crosswind simulation

Horizontal dispersion for double engine simulation after 1 second

Horizontal dispersion for double engine simulation (left engine) after 1 second
Vertical dispersion:

Headwind simulation

Crosswind simulation

Vertical dispersion for double engine simulation (left engine) after 1 second
APPENDIX C

Double engine analysis of plume dispersion after 10 seconds

Horizontal dispersion:

Headwind simulation

Crosswind simulation

Horizontal dispersion for double engine simulation after 10 seconds

Horizontal dispersion for double engine simulation after 10 seconds
Vertical dispersion:

Headwind simulation

Crosswind simulation

Vertical dispersion for double engine simulation after 10 seconds
Double engine analysis of plume dispersion after 42 seconds

Horizontal dispersion:

![Headwind Simulation](image)

![Crosswind simulation](image)

Horizontal dispersion for double engine simulation after 42 seconds
Vertical dispersion:

Headwind Simulation

Crosswind simulation

Vertical dispersion for double engine simulation after 42 seconds
Headwind simulation

Crosswind simulation

Iso-surface of 1 ppm with ground contour plot
APPENDIX E

Iso-surface of NOx concentration after 42 seconds of dispersion behind a double engine

Headwind simulation

Crosswind simulation
(This page intentionally blank)
For more information about the EEC Society, Environment and Economy Research Area please contact:

Ted Elliff
SEE Research Area Manager,
EUROCONTROL Experimental Centre
BP15, Centre de Bois des Bordes
91222 BRETIIGNY SUR ORGE CEDEX
France
Tel: +33 1 69 88 73 36
Fax: +33 1 69 88 72 11
E-Mail: Ted.Elliff@eurocontrol.int

or visit: http://www.eurocontrol.int/