ALAES
CFD Comparison of Buoyant and Non-Buoyant Turbulent Jets

EEC/SEE/2007/002
ALAQS CFD Comparison of Buoyant and Non-Buoyant Turbulent Jets

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

Author(s):
Syoginus ALOYSIUS, Luiz C. WROBEL

School of Engineering & Design
Brunel University
Uxbridge
Middlesex
UB8 3PH
UK
Web: http://www.brunel.ac.uk/about/acad/sed

Ian Fuller, EUROCONTROL Experimental Centre

Review:
ALAQS Peer Review Group.


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 compare the results of CFD simulations of buoyant and non-buoyant turbulent jets, and represents a step towards a better understanding of the source dynamics behind airplane jets to provide information for future improvements of existing dispersion models. The non-buoyant turbulent jet was first used to validate the CFD code with existing experimental and analytical results. The same geometry was then analysed with added buoyancy to highlight some of the properties that characterise plume rise behind an engine.

The method employed to characterise the plume dynamics adopts 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.

After the successful validation of the non-buoyant jet, the comparison emphasises the importance of two parameters: the turbulence intensity at the exhaust and the vortical structures. They are both interconnected in the definition of the potential core length, which was found to be shorter for the buoyant turbulent jet.

Three regimes govern the buoyant jet, namely the jet, intermediate and plume regimes. The non-buoyant results exhibit only the jet regime.

The next step in this research will include ground effects. By progressively reducing the distance between the jet axis and the ground, important plume dispersion information can be collected which will benefit other dispersion models.
This report aims to compare the results of CFD simulations of buoyant and non-buoyant turbulent jets, and represents a step towards a better understanding of the source dynamics behind airplane jets to provide information for future improvements of existing dispersion models. The non-buoyant turbulent jet was first used to validate the CFD code with existing experimental and analytical results. The same geometry was then analysed with added buoyancy to highlight some of the properties that characterise plume rise behind an engine.
(This page intentionally blank)
# ALAQS CFD Comparison of Buoyant and Non-Buoyant Turbulent Jets

## TABLE OF CONTENTS

1 **INTRODUCTION** ........................................................................................................................................................................... 1

2 **Forthmann Experiment Validation** .................................................................................................................................................. 2
   2.1 CFD simulation........................................................................................................................................................................... 2
   2.2 Analysis of the Results................................................................................................................................................................. 5
      2.2.1 Axial Velocity Profile for Non-Buoyant Jet......................................................................................................................... 5
      2.2.2 Vertical Velocity Profile for Non-Buoyant Jet...................................................................................................................... 8
      2.2.3 Theoretical Analysis ......................................................................................................................................................... 9
         2.2.3.1 Tollmien and Goertler theories.......................................................................................................................... 10
         2.2.3.2 Tollmien solution................................................................................................................................................ 10
         2.2.3.3 Goertler solution.................................................................................................................................................. 12
         2.2.3.4 Some comments on the theoretical approaches .................................................................................................. 13
         2.2.3.5 Comparison of theoretical and CFD results ........................................................................................................... 14
      2.2.4 Some Problems with Forthmann’s Experiment ............................................................................................................. 17
   2.3 Buoyant Jets .............................................................................................................................................................................. 18
      2.3.1 Axial Velocity Profile for Buoyant Jet.......................................................................................................................... 18
      2.3.2 Vertical Velocity Profile for Buoyant Jet......................................................................................................................... 20

3 **Three-dimensional study of free turbulent jet** ............................................................................................................................. 22
   3.1 Geometry and Boundary Conditions ........................................................................................................................................ 23
   3.2 Mean Velocity Profile Comparison ........................................................................................................................................ 23
   3.3 Vorticity Profile Comparison ............................................................................................................................................. 28

4 **CONCLUSIONS** ......................................................................................................................................................................... 31

5 **REFERENCES** ........................................................................................................................................................................ 32
LIST OF FIGURES

FIGURE 1 - GENERAL DESCRIPTION OF PLANE JET ................................................................. 3
FIGURE 2 - MESH SENSITIVITY RESULTS .................................................................................. 4
FIGURE 3 - TWO-DIMENSIONAL JET VELOCITY CONTOURS .................................................. 5
FIGURE 4 - VELOCITY PROFILE ALONG THE AXIS OF THE JET ............................................. 6
FIGURE 5 - TURBULENCE KINETIC ENERGY PROFILE ALONG THE JET AXIS ....................... 6
FIGURE 6 - VELOCITY PROFILE COMPARISON ALONG THE CENTRE OF THE JET ............. 7
FIGURE 7 - POTENTIAL CORE DESCRIPTION ........................................................................... 8
FIGURE 8 - CFD VERTICAL VELOCITY PROFILE COMPARISON AT DIFFERENT DISTANCES BEHIND THE JET .................................................................................. 9
FIGURE 9 - TOLLMIEN VERTICAL VELOCITY PROFILE ................................................................ 12
FIGURE 10 - GOERTLER VERTICAL VELOCITY PROFILE .......................................................... 13
FIGURE 11 - VERTICAL VELOCITY COMPARISON BETWEEN CFD AND THEORETICAL METHODS .................................................................................................................... 14
FIGURE 12 - ANGLE OF SPREAD COMPARISON ABOVE THE CENTRELINE ................................ 15
FIGURE 13 - TURBULENCE KINETIC ENERGY COMPARISON AT DIFFERENT DISTANCES BEHIND THE JET .............................................................................................. 16
FIGURE 14 - VORTICITY CONTOUR AROUND Z-AXIS WITH ISOLINES (1/S) .............................. 16
FIGURE 15 - VORTICITY PROFILE COMPARISON AT DIFFERENT DISTANCES BEHIND JET .......................... 17
FIGURE 16 - HORIZONTAL VELOCITY PROFILES WITH AND WITHOUT BUOYANCY ............... 18
FIGURE 17 - VELOCITY PROFILE AT DIFFERENT DISTANCES ABOVE AND BELOW THE JET AXIS ................................................................. 19
FIGURE 18 - VELOCITY PROFILES FOR BUOYANT JET ............................................................. 20
FIGURE 19 - VELOCITY PROFILE COMPARISON WITH BUOYANCY AT DIFFERENT DISTANCES BEHIND THE JET ..................................................................................... 20
FIGURE 20 - ANGLE OF SPREAD COMPARISON WITH BUOYANCY JET ABOVE THE CENTRELINE ..... 21
FIGURE 21 - ANGLE OF SPREAD TREND LINE WITH ADDITION OF BUOYANCY ...................... 21
FIGURE 22 - VORTICITY PROFILE COMPARISON AT DIFFERENT DISTANCES BEHIND BUOYANT JET .................................................................................................................. 22
FIGURE 23 - GEOMETRY AND MESH DISTRIBUTION OF THE CONTROL VOLUME ................. 23
FIGURE 24 - CENTRELINE AXIAL VELOCITY COMPARISON BETWEEN BUOYANT AND NON BUOYANT JET AT DIFFERENT TIME STEPS ....................................................................... 24
FIGURE 25 - MEAN VELOCITY PROFILE EVOLUTION FOR NON-BUOYANT AND BUOYANT JETS .............................................................................................................................. 25
FIGURE 26 - ANGLE OF SPREAD COMPARISON AFTER 1S ABOVE THE CENTRE OF THE JET AXIS .......................................................................................................................... 26
FIGURE 27 - ANGLE OF SPREAD COMPARISON AFTER 5S ABOVE THE CENTRE OF THE JET AXIS ........................................................................................................................ 27
FIGURE 28 - ANGLE OF SPREAD COMPARISON AFTER 10S ABOVE THE CENTRE OF THE JET AXIS ......................................................................................................................... 28
FIGURE 29 - EVOLUTION OF THE MAXIMUM HEIGHT OF THE PLUME CENTRELINE ............. 28
FIGURE 30 - INSTANTANEOUS VELOCITY PROFILE AFTER 5S FOR THE BUOYANT JET ............. 29
FIGURE 31 - VORTICITY MAGNITUDE AT 5S FOR THE BUOYANT JET ........................................ 29
FIGURE 32 - Z-VORTICITY COMPARISON AFTER 5S ................................................................... 30
# ABBREVIATIONS

<table>
<thead>
<tr>
<th>Abbreviation</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>CFD</td>
<td>Computational Fluid Dynamics</td>
</tr>
<tr>
<td>BS</td>
<td>Bell Shape</td>
</tr>
<tr>
<td>SR</td>
<td>Successive Ratio</td>
</tr>
<tr>
<td>RMS</td>
<td>Root Mean Square</td>
</tr>
<tr>
<td>TI</td>
<td>Turbulence Intensity</td>
</tr>
<tr>
<td>LS</td>
<td>Length Scale</td>
</tr>
<tr>
<td>ALAQS</td>
<td>Airport Local Air Quality Studies</td>
</tr>
<tr>
<td>NOx</td>
<td>Nitrogen Oxides</td>
</tr>
<tr>
<td>LES</td>
<td>Large Eddy Simulation</td>
</tr>
<tr>
<td>PIV</td>
<td>Particle Image Velocimetry</td>
</tr>
</tbody>
</table>
1 INTRODUCTION

The study of horizontal turbulent jets has been an important and interesting topic for many years. An early experimental investigation was conducted by Forthmann (1934) on a plane turbulent jet [Ref 1.]. Since then, progress has been very slow and it is only in the last decade that extensive research has been conducted on the flow structure of turbulent jets [Ref 2.]. This sudden interest in the field is due to its wide application in science and engineering. For instance, an airplane jet engine emits different types of pollutants. The study of their dispersion will help dispersion modellers to produce more accurate simulations and predictions to regulate airport traffic.

Experimental studies on turbulent flows have increased in accuracy from simple pitot tube measurements to complex Particle Image Velocimetry (PIV) photos, but this can be very expensive and time consuming for such complex problems. Computational Fluid Dynamics (CFD) has become an alternative to those complicated and laborious works, especially with the increase in computer technology. Processor speeds have increased at an unprecedented rate and models such as Large Eddy Simulation (LES) can play a major role in understanding the flow behaviour.

The basic concept of the LES model is to explicitly solve the larger eddies of the control volume, whereas the smaller eddies are modelled through spatial filtering. Although this model is not suited for a complete airport simulation at present due to extensive requirements of computer speed and memory, it can be used efficiently in local studies related to source dynamics and engine plume/wing tip vortex interactions.

A LES model was used in this report to investigate the differences between turbulent buoyant and non-buoyant jets in a free atmosphere condition, highlighting the dispersion mechanism behind the exhaust. The first part of the report aims at validating the CFD simulation with experimental work in a non-buoyant condition. Analytical results are also derived and compared to the simulation. Buoyancy effects were then included in the form of a passive species at different density and temperature from the ambient air.
2 FORTHMANN EXPERIMENT VALIDATION

This experiment, conducted in 1934, is the earliest on plane turbulent jets recalled through the literature [Ref 1.]. The simplicity of its results and the comparison with different analytical models makes it an ideal choice to validate CFD models.

Forthmann used a rectangular jet nozzle 0.03m high and 0.65m wide, i.e. with an aspect ratio of 21.7, to investigate the free jet expansion. The jet releases an air stream at a velocity of 35m/s into an unbounded ambient still air. The measurements were taken by a pitot tube which records static pressure of the flow, and the final velocity was computed by using an equation for the dynamic pressure [Ref 3.].

2.1 CFD simulation

An extensive research has been undertaken to find the best way to represent the overall geometry of Forthmann's experiment, and to conduct the CFD calculations more efficiently. In his paper, Forthmann suggested his free jet experiment could be reduced to a two-dimensional problem, and compared his results with analytical ones based on the two-dimensionality approach of the problem. This approach will be discussed in more detail later in the report.

While reducing the geometry to two dimensions, it is important to find out the best control volume domain to enclose the flow. This characterisation is closely linked with the boundary conditions to be assigned on each wall.

The boundary conditions were setup as follows:

- The inlet jet has a velocity of 35m/s. The inlet condition is assigned right at the beginning of the control volume in accordance with the findings of Klein et al. [Ref 4.] and Stanley et al. [Ref 5.], also reported by Celik et al. [Ref 6.]. The air coming out of the jet is isothermal with the ambient air of the control volume, so that no buoyancy is present in the simulation.
- The boundaries just next to the jet were set as walls.
- The top and bottom boundaries were also assigned as walls.
- The classical outflow boundary condition is assigned at the outlet.

A simple preliminary study confirmed that a control volume of dimensions 4.4m by 3.2m is sufficiently large to avoid influences from the artificial boundaries. A proper meshing can now be applied to the domain, which is mostly refined near the jet exhaust up to a distance of 1m behind the jet. A mesh sensitivity analysis was then conducted, comprising of three tests with different mesh density by varying the number of grid points and the grid spacing near the jet. The figure and table below show the mesh distribution for each test.

It is known from Forthmann’s results that the core of the jet geometry has a bell shape (B.S.) distribution profile. The other edges have a simple successive ratio (S.R.) grading, where the length of the following edge is multiplied by a constant ratio.
The results presented in Figure 2 are for the velocity profile at a location 0.2m behind the jet for different tests. The graph is non-dimensionalised as follows:

- The non-dimensional coordinate \( \frac{y}{b} \) represents the ratio between vertical position and distance from the axis to the point at which the velocity is equal to \( \frac{1}{2} u_m \).

- The non-dimensional velocity \( \frac{u}{u_m} \) represents the ratio between local velocity and jet axis velocity.
The influence of mesh density on the results can be better seen near the centre of the jet and on the sides of the curve where Test 1 is slightly more pronounced than Test 2, which shows also some deviation from Test 3 which has a higher mesh density. Thus, the results for Test 3 were taken for comparison with Forthmann’s experimental work.

Because of the lack of experimental information, the CFD simulation was run long enough to reach a steady state. Another important parameter not mentioned in Forthmann's paper is the level of turbulence coming out of the jet exhaust. The setting of the turbulence parameters includes the turbulence intensity and its length scale. Wygnanski and Fiedler [Ref 7.] reported in their study of self-preserving jets a turbulence intensity value of 1%, whereas Deo et al. [Ref 8.] obtained a value of 0.5%. Reviewing some of the experimental work on jets, Rodi [Ref 9.] also found a variety of turbulence intensities up to 1%, although some refer to other types of jets.

A study was then conducted to observe the data sensitivity when the turbulence intensity and length scale parameters are modified.

1. Turbulence intensity

Turbulence intensity is obtained by dividing the RMS value of the velocity component by its mean value [Ref 10.]. Normally, wind tunnel experiments have very low turbulence intensity but in real conditions such as atmospheric studies this value can reach up to 40% in a suburban area near the ground [Ref 11.].

Appendix A shows that, even though the differences are very low, the higher the turbulence intensity the more symmetrical is the predicted behaviour of the jet profile. Moreover, in the near field very close to the jet, Forthmann’s experiment suggests a bell shape for the velocity profile in the fully developed region. It can be seen that this shape is very well reproduced with a turbulence intensity of 5%, whereas the other two results show a portion of the potential core which still appear to be in the flow development region. This point will be discussed in greater detail further on the report, but for now it appears that the length of the development region is very much influenced by the turbulence intensity. The lower the turbulence intensity, the farther the flow will take to fully develop.

2. Length scale

The length scale is related to the size of the large eddies which contain the energy in the turbulent flow [Ref 10.]. Appendix B shows that there is no influence from the turbulence length scale for a range of values between 0.025m and 1m.
2.2 Analysis of the Results

Figure 3 presents the jet velocity contours. The range of velocity isolines in this plot goes from 0 to a maximum of 35m/s, which is the jet velocity exhaust. The jet is clearly symmetrical, with its centre at the axis of symmetry.

The isolines can be compared with Abramovich's representation of equal velocity lines in submerged jets [Ref 12.]. The above plot fits very closely to his findings, with the triangular potential core next to the jet exhaust and the rounded shape of isolines as the flow progresses through the control volume.

The comparison of velocity contour plots with theoretical equations will provide a better quantitative assessment of the accuracy of the CFD simulations. This process will be discussed in two different sections, for the axial and vertical velocities.

2.2.1 Axial Velocity Profile for Non-Buoyant Jet

Figure 3 clearly highlighted the velocity decay as the flow progresses behind the jet. This can be further verified in Figure 4. In the abscissa, the distance behind the jet is multiplied by an experimental factor $a (=0.11$ for Forthmann’s experiment, as explained further in the report) and divided by the half-length of the jet opening (0.015m), while the ordinate is the ratio of the local velocity to the maximum axial velocity.

The region of maximum velocity near the jet corresponds to the potential core distance up to a point where the velocity gradually decays asymptotically. Rajaratnam [Ref 1.] explained the disappearance of the potential core as a result of turbulence penetration, leading to the fully developed region. The jet entrains the ambient fluid surrounding it, triggering an enhancement of turbulence. Because the jet is still strong near the exhaust, its influence is not perceptible but as it goes along the axis, its intensity increases leading to the velocity decay.
Figure 4 - Velocity profile along the axis of the jet

Figure 5 shows the turbulence kinetic energy profile along the jet axis. The start of the rise in turbulence corresponds approximately to the point where the velocity starts to decrease in the axial velocity profile. The turbulence intensity increases abruptly to a maximum followed by a gradual decay. The explanation of this behaviour will be discussed in greater detail further in the report, but at this point it is possible to say that this sudden rise of turbulence kinetic energy is due to the outer region vortices colliding in the centreline axis.

![Figure 5 - Turbulence kinetic energy profile along the jet axis](image)

It was found from the CFD centreline velocity that the length of the potential core extends to approximately 0.167m behind the jet. According to Rajaratnam [Ref 1.], this length can be calculated with the empirical formula:

$$\overline{x_0} = b_0 / \tan 4.8^\circ = 11.91^\circ b_0$$

For the current data, the formula gives a value of 0.179m.

The main assumption behind the above formula is that the angle $\alpha_i = 4.8^\circ$, which is constant in Rajaratnam’s case. It was found in the CFD simulations that the effect of the turbulence intensity changes the length of the potential core and, as a consequence, the angle $\alpha_i$. 
This statement is in line with the findings of Flora & Goldschmidt [Ref 13.], who found two different origins of similarity in their study of a turbulent free jet. The first is based on the widening of the jet (which will be discussed in detail later in the report), while the other is based on the decay of the axial velocity. They found that, at different Reynolds numbers, the locations of these virtual origins are independent of the shape of the section, but they are strongly dependent on the turbulence intensity of the jet [Ref 13.]. An increase in turbulence intensity at the inlet shifts the point of virtual origin upstream, well beyond the point where the jet is supposed to exhaust [Ref 13.]. This clearly shows that the length of the potential core in a plane jet is not fixed, but can change depending on the turbulence intensity.

Abramovich [Ref 12.], on the other hand, obtained empirical formulae for the axial velocity based on some experimental results. He showed that the velocity decay for a round jet with constant release is of the form \( u_m = \frac{\text{const}}{x} \), with \( u_m \) the centreline velocity and \( x \) the distance along the jet axis, while the decay for a plane jet is of the form \( u_m = \frac{\text{const}}{\sqrt{x}} \). The velocity decay expressions for different types of problems are given by Schlichting [Ref 16.].

The constant in the above expressions can be found experimentally. For a plane jet, the final formula for the axial velocity profile is

\[
\frac{u_m}{u_0} = \frac{1.2}{\sqrt{\frac{ax}{b_0}}}
\]

with \( u_0 \) the jet velocity and \( b_0 \) the diameter of the outer boundary of the jet. The constant \( a \) is found by trial and error; for Forthmann’s experiment, a value of \( a = 0.11 \) provided the best fit to his curve.

Figure 6 - Velocity profile comparison along the centre of the jet

Figure 6 shows the jet axial velocity profile. In addition to the curve presented in Figure 4, the plots include two CFD results with different inlet turbulence intensities and the profile obtained with Abramovich’s formula. The curves are in good agreement far behind the jet, all showing asymptotic velocity decay. The slight differences come right near the inlet of the jet.
The effect of turbulence intensity is clearly seen when comparing the CFD simulations. The potential core is much longer for low turbulence intensity; this is accordance with the previous statement. The lower the turbulence intensity, the farther the flow will take to fully develop.

At first glance, the plot of the CFD simulation with 5% turbulence intensity at its inlet might seem wrong compared to the analytical plot; moreover, the potential core also looks longer (0.1975 m) than the CFD result. The reason for such difference is that the theoretical approach does not take into account the transitional region between the initial and main regions of the jet.

Abramovich [Ref 12.] corrected this difference by showing a plot very close to the CFD one, and found that for a plane jet the length of the potential core is \( x_0 = \frac{x_n}{1.5} \) whereas for an axially symmetrical jet \( x_0 = \frac{x_n}{1.2} \), with \( x_n \) the distance to the end of the transition part as shown in Figure 7.

\[
\frac{u_m}{u_0} = x \frac{x_n}{x_0}
\]

This correction finally gives a theoretical value of 0.1316 m for the potential core length. The CFD result of 0.167 m is located between the theoretical prediction of Abramovich [Ref 12.] and the empirical formula of Rajaratnam [Ref 1.] (0.1316<0.167<0.17865).

### 2.2.2 Vertical Velocity Profile for Non-Buoyant Jet

One of the major characteristics of a plane jet is its capability to show a self-preserving behaviour after the flow development region. This is often known as exhibiting a self-similar velocity pattern.

Bradbury & Riley [Ref 14.] defined a flow as self-preserving when the velocity along its centreline is much greater than the free-stream velocity. Rodi [Ref 9.], on the other hand, in his review of experimental data, defined a flow as self preserved when “one velocity and one length scale are sufficient to render its time averaged quantities dimensionless functions of one geometrical variable only”.

Abramovich [Ref 12.] explained the reason of the self-similar pattern found in experimental works with the absence of solid boundaries near the flow. This is specifically applied to turbulent free jets. No solid boundaries mean that no laminar sub-layer is present and, as a consequence, viscosity can be neglected [Ref 12.].
Rodi [Ref 9.] explained the mathematical implications of this by quite considerably reducing his two-dimensional equations for his calculation of the turbulent shear stress \( \bar{\mu} \bar{\nu} \) and the turbulent kinetic energy \( TKE \) by stating that all convection, diffusion, generation, pressure transport and dissipation terms have constant ratios, so that both energy and shear stress balance are in dynamic equilibrium [Ref 9.]. This statement is supported by Wygnanski & Fiedler [Ref 7.], who achieved a self-preserving state when all the turbulence components of their experimental data were in an equilibrium state.

Taking into account what was discussed previously, the dimensionless functions are setup as described in the previous section with the abscissa dividing the distance away from the jet axis by the distance where the velocity is half the axial velocity, and the ordinate showing the ratio of the local velocity to the axial velocity.

![Figure 8 - CFD vertical velocity profile comparison at different distances behind the jet](image)

Figure 8 shows that all plotted lines at different distances behind the jet have approximately the same shape. Squire & Trouncer [Ref 15.] compared this profile to a cosine curve, and most of the results on plane jets found in the literature follow this trend.

The profile also shows a symmetrical velocity with respect to the jet axis. The highest velocity is situated on the centreline, as discussed earlier on (see also [Ref 14.]).

Figure 8 shows some small deviations further away from the centre of the jet but the error is minimal. It will be shown later on the report that even the theory disagree on the representation of the self-similarity away from the centre of the jet axis.

### 2.2.3 Theoretical Analysis

There are different theoretical models for the study of turbulent free jets. Abramovich [Ref 12.] recalled and analysed four of them, namely Prandtl’s old and new theories, Taylor’s theory and Reichardt’s theory. In this report, only Prandtl’s old and new theories will be discussed and compared to the results found with the CFD simulation.

There are two distinctive approaches for solving the Prandtl’s old theory. The first is based on the mixing length hypothesis (Prandtl-Tollmien), while the other is based on the assumption that there is a constant eddy exchange coefficient across the flow (Prandtl-Schlichting) [Ref 7.]. Both theories were compared with experimental investigations made by respective authors, confirming that Prandtl’s old theory can be used for the study of turbulent free jets [Ref 12.].
The difference between the old and new theories lies in the assumption that, in the old theory, the mixing length was kept constant whereas in the new theory the turbulent viscosity is constant along the jet [Ref 16.].

2.2.3.1 Tollmien and Goertler theories
As previously mentioned, the problem can be simplified as a two-dimensional jet releasing isothermal air with a velocity of 35m/s in a stagnant surrounding. The conservation laws can then be reduced to two, namely mass and momentum because of the isothermal assumption, as follows:

Continuity equation:

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0$$

Momentum equation:

$$u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} = \frac{1}{\rho} \frac{\partial }{\partial y} ( \rho \tau_{i} )$$

The main differences between the Tollmien and the Goertler models will be discussed next.

2.2.3.2 Tollmien solution
The Tollmien theoretical approach is based on the Prandtl mixing length formula [Ref 16.]:

$$\tau_{i} = \rho l^{2} \left( \frac{\partial u}{\partial y} \right)^{2}$$

where \( l \) is the mixing length. This additional equation is of great importance since it enables to find the three unknowns \( u, v \) and \( \tau_{i} \).

First, the turbulent shear stress can be written in a different manner, assuming that the mixing length is proportional to the distance where the velocity is half the axial velocity \( \bar{b} \), so that \( \beta \times \bar{b} \) or \( \beta \times C_{2} \times x \), with \( \beta \) a constant and \( C_{2} \) a coefficient found experimentally [Ref 1.]. This allows the momentum equation to be written as follows:

$$\frac{1}{\rho} \frac{\partial }{\partial y} ( \rho \beta^{2} C_{2}^{2} x^{3} \left( \frac{\partial u}{\partial y} \right)^{2} ) = a^{3} x^{2} \frac{\partial u}{\partial y} \frac{\partial^{2} u}{\partial y^{2}}$$

with \( a^{3} = 2(\beta C_{2})^{2} \).

The continuity equation can be evaluated as follows:

$$\frac{u}{u_{m}} = f \left( \frac{y}{ax} \right) = f(\phi)$$

with \( \phi = \frac{y}{ax} \).

It was shown earlier in the report that \( u_{m} = C_{1} U_{0} \sqrt{\frac{b_{0}}{x}} = \frac{n}{\sqrt{x}} \) with \( n \) constant, so \( u = \frac{n}{\sqrt{x}} \times f \).
Let \( u = \frac{\partial \psi}{\partial y} \) and \( v = -\frac{\partial \psi}{\partial x} \), where \( \psi \) is a stream function. Integrating \( u \) will give

\[
\psi = \frac{a}{\sqrt{x}} \int f \times a \times x \, d\phi = a \frac{n}{\sqrt{x}} \int f \, d\phi
\]

Let \( F = \int f \, d\phi \) and replacing in the definition of \( v \) results in

\[
v = -\frac{\partial \psi}{\partial x} = -\frac{\partial}{\partial x} \left( a n \sqrt{x} F \right) = -a n \left( \frac{F}{2 \sqrt{x}} - \sqrt{x} F' \phi \right) = a n \left( \phi F' - \frac{F}{2} \right)
\]

where \( F' \) is the derivative of \( F \) with respect to \( \phi \).

Substituting \( u = \frac{n}{\sqrt{x}} f \) into \( \frac{\partial u}{\partial x} \) gives \( \frac{\partial}{\partial x} \left( \frac{n}{\sqrt{x}} F' \right) \), because \( F = \int f \, d\phi \) and \( f = F' \).

Differentiating with respect to \( x \) now gives

\[
\frac{\partial u}{\partial x} = -\frac{n}{x \sqrt{x}} \left( \frac{F'}{2} + \phi F^{*} \right)
\]

and so

\[
u \frac{\partial u}{\partial x} = -\frac{n^{2}}{x^{2}} \left( \frac{F'^{2}}{2} + \phi F^{*} \right)
\]

In the same manner, it is possible to find

\[
\frac{\partial u}{\partial y} = \frac{\partial}{\partial y} \left( \frac{n}{\sqrt{x}} \phi F' - \frac{F}{2} \right) = \frac{n}{a \sqrt{x}} F^{*}
\]

so that

\[
\frac{\partial u}{\partial y} = \left[ a n \left( \phi F' - \frac{F}{2} \right) \right] \times \left[ \frac{n}{a \sqrt{x}} F^{*} \right] = \frac{n^{2}}{x^{2}} \left[ \phi F^{*} F^{*} - \frac{F F^{*}}{2} \right]
\]

and

\[
1 \frac{\partial}{\partial y} = a^{3} x^{2} \frac{\partial u}{\partial y^{2}} = \frac{n^{2}}{x^{2}} F^{*} F^{*}
\]

Replacing all the underlined equations in the momentum equation gives:

\[
-\left( \frac{1}{2} F'^{2} + \phi F^{*} \right) + \left( \phi F^{*} - \frac{1}{2} F^{*} \right) = F^{*} F^{*} \Rightarrow 2 F^{*} F^{*} + F F^{*} + F'^{2} = 0 \Rightarrow 2 F^{*} F^{*} + \frac{d}{d\phi} (F F') = 0
\]

Integrating the above equation gives \( F'^{2} + F F' = C \). The constant \( C \) can be evaluated from the boundary conditions, and was found to have a zero value. The final expression is \( F'^{2} + F F' = 0 \), which is a non-linear second-order ordinary differential equation. Tollmien was the first to solve it in 1926, and his curve is presented in Figure 9.
Wygnanski & Fiedler [Ref 7.] compared the Tollmien curve and the one derived by Schlichting to their experiment, but found that neither of the two profiles agreed well with their experimental data across the flow. They suggested that the Tollmien solution is mostly suited in the outer part of the flow, whereas the Schlichting curve fits better the inner part of the flow.

2.2.3.3 Goertler solution

Goertler used the second equation of Prandtl [Ref 16.] because the definition of the kinematic eddy viscosity $\varepsilon$ in Prandtl’s old theory was not properly modelled. As a matter of fact, the equation $\varepsilon = \frac{1}{l^2} \frac{du}{dy}$ vanishes at the points where $\frac{du}{dy}$ is equal to zero. This behaviour was proved wrong in the experimental work of Reichardt [Ref 12.]. Prandtl had to establish another equation to correct the kinematic viscosity effect [Ref 16.]. The Prandtl turbulent shear stress was then given as $\tau_i = \rho \times \varepsilon \times \frac{\partial u}{\partial y}$. Similar to the Tollmien approach, $\varepsilon$ is proportional to the distance where the velocity is half the axial velocity, that is, $\varepsilon = k u_m b$, where $k$ is constant.

Let

$$\xi = \sigma \frac{y}{x} \Rightarrow \frac{u}{u_m} = F\left(\sigma \frac{y}{x}\right) = F'(\xi)$$

Previously, it was found that $u_m = \frac{n}{\sqrt{x}}$, so $u = \frac{n}{\sqrt{x}} F'(\xi)$. The same procedure as Tollmien’s is adopted to find that $\psi = \frac{n}{\sigma} \sqrt{x} F'(\xi)$. Substituting into $-\frac{\partial \psi}{\partial x}$, it is possible to find

$$v = \frac{n}{\sigma} \frac{1}{\sqrt{x}} \left(\xi F' - \frac{F}{2}\right)$$

and $\tau_i = \rho k C_2 \frac{n^2}{x} \sigma F''$. Replacing this result in the momentum equation gives:

$$\frac{1}{2} FF' + k C_2 \sigma^2 F'' = 0$$
Calling \( \sigma = \frac{1}{2\sqrt{kC_2}} \), the equation differential equation then becomes

\[
2FF' + F'' = 0
\]

By integrating and applying the boundary conditions, Goertler found in 1942 that the equation can be simplified to

\[
F^2 + F' = 1
\]

with \( F = \tanh(\xi) = \frac{1 - e^{-2\xi}}{1 + e^{-2\xi}} \) and \( F' = 1 - \tanh^2(\xi) \). This function is represented in Figure 10.

![Goertler vertical velocity profile](image)

**Figure 10 - Goertler vertical velocity profile**

Schlichting [Ref 16.] compared Goertler’s velocity distribution with the experimental work of Reichardt, and found a very good agreement between the two.

### 2.2.3.4 Some comments on the theoretical approaches

Abramovich [Ref 12.] compared both Tollmien’s and Goertler’s theories with the experimental data of Reichardt. He found differences between the two theoretical approaches but, similar to Wygnanski & Fiedler, pointed out that the Tollmien curve best fits the experimental data on the outer part of the flow. Zijnen [Ref 17.] confirmed this remark by comparing his experimental data with the two theoretical approaches in question, and also found that the Tollmien solution better represents the experimental curve on the outer edge whereas the Goertler theoretical profile matches more closely near the jet axis. Zijnen also found that his experimental velocity profile could be represented by a Gaussian curve with the equation [Ref 17.]:

\[
\frac{u}{u_m} = \exp \left( -0.693 \left( \frac{y}{b} \right)^2 \right)
\]

There are many variations of the above Gaussian profile for turbulent plane jet flows in the literature, and also for other similar flows such as circular jets and co-flows, but it is not the objective of this report to review all of them. Consequently, only Zijnen’s Gaussian profile will be compared to the CFD simulation.
2.2.3.5 Comparison of theoretical and CFD results

The graph presented in Appendix C shows a comparison between the Tollmien’s and Goertler’s analytical results and Zijnen’s Gaussian curve. The Gaussian curve matches very closely the Goertler profile in the inner part of the flow, whereas in the outer part it shows a good fit to the Tollmien profile. Figure 11 adds the CFD simulation plots at different distances behind the jet to the theoretical and Gaussian curves.

![Figure 11 - Vertical velocity comparison between CFD and theoretical methods](image)

The CFD results follow the Gaussian curve in that they match the Goertler profile in the inner part of the flow and the Tollmien curve in the outer part of the flow, up to a position around ±1.5 in the dimensionless abscissa axis, when the plots begin to separate. An important point to note is that all the CFD results lie between the two theoretical curves. Further away from ±2.5, it can be seen that the velocity ratio does not reach zero like the Tollmien profile but stays between this and the Goertler curve (0.05), although they converge to zero if the horizontal graph scale were extended.

Wygnanski & Fiedler [Ref 7.] discussed this phenomenon and found out that the changes in velocities and velocity fluctuations are more pronounced in the vertical direction than in the streamwise direction, with a substantial scatter at the edges of the flow.

Forstall & Shapiro [Ref 18.] reported some difficulties in finding the point where the velocity reaches zero, and concluded that this was due to the “outer boundary mixing” with its surrounding. They also emphasized the importance of the spreading parameter in the study of jets.

An earlier report by Flora & Goldsmidth [Ref 13.] stated that distance of the virtual origin where the spread line crosses the axis of the jet is highly dependent on the turbulence intensity. It was also reported that the turbulence intensity affects the potential core distance.

Kotsovinos [Ref 19.] pointed out that changes in turbulence intensity affect the position of the virtual origin. By analysing some experiments, he found that instead of using the distance where the velocity reaches zero, the distance where the velocity attained half the axial velocity ($b$) is of greater value. He also reported that the spreading growth of the jet can be considered as linear, but this is only valid up to 200 diameters behind the jet. He concluded that all the experimental work he analysed would fit a single curve with the equation [Ref 19.]:

$$\frac{b}{2b_0} = K_1 \left( \frac{x}{2b_0} + K_2 \right)$$
where $2b_0$ is the thickness (diameter), and $K_1$ and $K_2$ are coefficients derived from experiments. From Forthmann’s experiment, he established the values $K_1 = 0.096$ and $K_2 = 0.6$ [Ref 19].

By correlating each experiment, Kotsovinos derived a unique third-order polynomial equation [Ref 19]:

$$\frac{b}{2b_0} = 0.228 + 0.0913\left(\frac{x}{2b_0}\right) + 0.00005101\left(\frac{x}{2b_0}\right)^2 + 0.000000331\left(\frac{x}{2b_0}\right)^3$$

which is only valid up to 200 diameters behind the jet. Figure 12 presents a correlation between Kotsovinos’s curve with Forthmann’s and CFD plots, using $K_1 = 0.096$ and $K_2 = 0.6$.

![Figure 12 - Angle of spread comparison above the centreline](image)

All three curves are in very good agreement; as expected. Forthmann’s and Kotsovinos’ results are the closest because Kotsovinos used Forthmann’s data to derive his equation. The slight difference concerning the CFD solution might have come from the way the parameter $b$ was found, since $b$ was interpolated from the velocity profile.

Kotsovinos realised that the plots are linear and a trend line can be found. Two important points can be drawn out of the resulting equations:

- Firstly, $\frac{K_2}{K_1}$ offers a glimpse of where the location of the virtual point is when $y = 0$. In this case, it is clear that the CFD virtual point is located between Kotsovinos’ and Forthmann’s locations. Indeed, Kotsovinos’ distance of the virtual point was found to be the farthest away, whereas Forthmann’s is the closest to the real point of the jet exhaust.

- Finally, $\frac{K_1}{2b_0}$ gives a measure of the spreading rate the jet is subject to. It can be seen that the CFD result shows a slightly higher spreading rate than the other two curves. The difference is about 4.7% to Forthmann’s experimental work and 5.5% to Kotsovinos’ empirical equation.

It is now possible to focus on the plume spreading by studying what Forstall and Shapiro called the “outer boundary mixing” with its surrounding [Ref 18].
To analyse the mixing process with the ambient surrounding, Figure 13 shows the turbulence kinetic energy across the vertical axis.

![Figure 13 - Turbulence kinetic energy comparison at different distances behind the jet](image1)

It can be seen from the above figure that the symmetrical pattern is conserved and there are two peaks of high turbulence on each side of the jet. An analysis of the centreline confirms that there is a sudden increase in turbulence kinetic energy up to 0.5m behind the jet and a slow asymptotic decrease after that point.

The position of the peaks with respect to the centreline distance increases as the flow travels away from the exhaust. This is a good representation of the widening of the jet spread, as discussed earlier. The position of the maximum turbulence is approximately the distance \((b)\) where the velocity is half of the axial velocity. Another observation is the reduction in the peak turbulence intensity as the flow progresses, due to the flow entrainment with its surroundings.

Townsend [Ref 20.] discussed how entrainment and rate of spread are interconnected. He introduced the concept of entrainment eddies in addition to the ones already present in the jet; these eddies are the causes of the velocity perturbations at the extremities of the jet [Ref 20.]. The development of vortices at these locations occurs due to the jet flow encountering the surrounding irrotational fluid. In the present study, the existence of vortices is discussed in the form of vorticity contour plots.

![Figure 14 - Vorticity contour around z-axis with isolines (1/s)](image2)
It can be seen from the above plot that vorticity is very high near the jet exhaust, and is mainly composed of counter-rotating vortices (positive at the top and negative at the bottom). The isolines show the symmetrical pattern of the vortices around the jet centreline. The following interesting points can be highlighted:

- The potential core is well represented with lower vortices inside it; this can be correlated with the turbulence intensity section discussed earlier.
- The boundaries of the jet spreading can be clearly seen as a separation between rotational and irrotational flows.

Figure 15 gives an insight into the vertical axis of the jet.

The vorticity magnitude is higher near the jet exhaust and gradually decreases as the flow evolves behind the jet. The location of the maximum vorticity amplitude also changes during this transport, going further away from the jet axis thus confirming the widening discussed earlier in the report.

Another interesting observation, related to the previous work of Aloysius et al. [Ref 21.] on CFD simulations of an aircraft engine during take-off, is how the counter-rotating vortices form and convey the jet momentum. Townsend [Ref 20.] reported that counter-rotating vortices exist on each side of the jet, but their evolution is more difficult to understand. Browne et al. [Ref 22.] also reported that the vortices undergo a “coalescence cascade”, meaning that instead of breaking up, they increase in size and merge into a single vortex further downstream. When this happens, a breakdown of the single vortex starts occurring and continues until there is nothing left of it far downstream [Ref 23.] (well away from this study’s length).

### 2.2.4 Some Problems with Forthmann’s Experiment

Rodi [Ref 9.] described three criteria for assessing experimental data:

1. The actual experimental flow must correspond closely to the ideal one.
2. The instruments and evaluation methods must be adequate.
3. Data must be consistent.

Forthmann’s experiment does not fall into any of the above categories.

First of all, the instruments used to conduct Forthmann’s experiment were quite basic. Only the streamwise velocity was recorded and from that, the self-similar behaviour was in some way
proven. Heskestad [Ref 23.] also criticised Forthmann's experimental apparatus and found an inadequacy in that the jet would not have reached its asymptotical state in the region where Forthmann carried out his experiment (between 0 to 25 diameters). In a similar study, Wygnanski [Ref 7.] found that to obtain a truly self-similar behaviour, measurements should be done after some 70 diameters from the nozzle otherwise the results would still exhibit considerable scatter at the edges of the flow, as found in the CFD simulation.

Unfortunately, no turbulence parameters were recorded because of the instrument used. The results of those parameters would have given a better knowledge and understanding of the free turbulence and the mechanism of turbulent diffusion, as suggested by Forstall & Shapiro [Ref 18.].

So far, the study has been limited to plane jets. As an airplane engine exhaust is circular, even though some of the properties of a plane jet would still be applied (self-similarity), a three-dimensional simulation of the a circular jet release into a co-flowing air stream with same density will also be carried out, adopting a similar jet dimension.

2.3 Buoyant Jets

Unfortunately there was no continuation of Forthmann’s experiments to include buoyancy, and no comparison can be made explicitly with experiments. This section initially aims to compare the results obtained in the previous non-buoyant configuration with a similar buoyant flow. Such condition is possible by adding a temperature difference and assigning a release of different species of different density. In that sense, a first test was made using the parameters of the CFM56-3C engine (velocity 80m/s, exhaust temperature 690K and mass fraction of passive NOx $0.74 \times 10^{-4}$), as this was the engine studied in a previous ALAQS report. It was found that the velocity and concentration were far greater and, as a consequence, the influences of the wall become significant.

One way to remedy this problem is to extend the computational domain, but this would not allow a comparison with previous results. The way adopted in this study was to keep the velocity at 35m/s but with a reduced concentration ($0.37 \times 10^{-3}$) to avoid the need for increasing the length of the control volume.

The simulation was run for the same time step as the non buoyant simulation in order to analyse the results.

2.3.1 Axial Velocity Profile for Buoyant Jet

![Figure 16 - Horizontal velocity profiles with and without buoyancy](image-url)
Figure 16 presents the velocity profile along the centre of the jet. It can be seen that all plots share a similar pattern near the jet exhaust; however, far downwind, the similitude still exists for the jet without buoyancy and the theory whereas the buoyant jet diverges completely from the other two.

The length of the potential core for the buoyant jet was 0.1458m. This length still lies between the two theoretical predictions (0.1316m<0.14578m<0.1786m). The figure shows that the CFD buoyant simulation follows closely the theoretical curve up to a distance of 0.3m behind the jet. It is only after that length that the buoyant jet curve clearly deviates from the other two plots.

Figure 17 shows the centreline velocity and four line plots at different distances above and below the jet exhaust. It can be seen that the velocities above the centreline increase close to the exhaust, whereas below the jet axis the increase in velocity is delayed. This sudden increase in velocity is due to the buoyancy acting on the flow behind the exhaust. It is also interesting to note that, below the centreline, the velocity increases slightly; this was caused by the jet spread reaching this height further downwind. To illustrate this and understand the buoyancy phenomenon, Figure 18 shows a contour plot of the jet streamwise velocity.

Figure 17 - Velocity profile at different distances above and below the jet axis

Figure 18 - Velocity contours for buoyant jet
Figure 18 shows that, unlike Figure 3 for a non-buoyant jet, the symmetrical pattern of the jet is now broken, as expected. The velocity above the jet axis is higher than below. The plume rise is much clearer as the flow progresses downwind.

The velocity pattern below the jet axis resembles the one for the non-buoyant jet, explaining the increase in velocity at 0.4m and 0.5m below the centreline. If the geometry were extended, the graph would have shown the rise and fall of velocity further away from the exhaust, as it will be demonstrated in the 3-D study section.

2.3.2 Vertical Velocity Profile for Buoyant Jet

Earlier in the report, it was shown that non-buoyant jets have a self-preserving behaviour behind the jet, away from the flow development region. The previous section showed that the symmetrical pattern is broken for the buoyant jet and, as a consequence, the point of maximum velocity will deviate from the jet centre.

This behaviour is somewhat shown in Figure 19, where the point of maximum velocity especially for the lines representing distances further away from the jet are deviated from the centreline.

Appendix D shows the CFD comparison between the buoyant and non-buoyant jets. The ordinate was changed to dimensional form to highlight the differences. It can be seen that the difference is much appreciable further behind the exhaust, with the non-buoyant jet showing a centred and symmetrical profile as discussed previously whereas the buoyant jet shows a completely different off-centred profile. The point of maximum velocity tends to the positive in the y direction; this means that the velocity is higher above the jet centreline, showing evidence of plume rise.

Another point that can be noted here is related to the extremities of the velocity profile. The outer boundary is the mixing region with the surrounding ambient. It can be seen that the velocity ratio at the upper extremity (above the centreline) is much higher and increases further downwind, whereas the lower extremity does not vary much and is much lower than the non-buoyant jet as the flow progresses downwind. This confirms again the rise of the plume.

In order to understand more clearly the rising of the plume, Figure 20 compares the jet spreading parameters, using Kotsovinos’ reference, by analysing different locations where the velocity attained half the axial velocity at different distances behind the jet. The figure clearly shows the plume rise and the different phases the jet undergoes as it progresses through the control volume.
Figure 20 - Angle of spread comparison with buoyancy jet above the centreline

Gebhart et al. [Ref 25.] reported three types of flow regions the jet encounters. This statement seems to be in line with the findings of Chen and Rodi [Ref 26.], who analysed and reviewed experimental work of an axisymmetric jet. In the first region, very close to the jet exhaust, they found that the flow behaviour replicated the pure jet, having an almost self-similar profile. The far field region is the furthest away from the exhaust; in this, they observed that the behaviour is more like a plume. Between these two regions is the intermediate region where the flow undergoes a transition from jet-like to plume-like behaviour.

Similar to the case of non-buoyant jets, a trend line can be drawn in the jet region to analyse Kotsovinos' parameters.

Figure 21 - Angle of spread trend line with addition of buoyancy

The jet region is governed by the trend line presented above; it is clear with the explanation given in section 2.2.3.5 that virtual point is situated behind the jet, similar to the non-buoyant case and Forthmann’s experiment. The value $\frac{K_2}{K_1}$ is very close to the CFD simulation without buoyancy and is still in between Forthmann’s experiment and Kotsovinos’ theoretical prediction for a non-buoyant jet.
The other parameter that can be analysed is the spreading rate \( \frac{K}{2h_0} \); once again, this value is very close to the CFD results without buoyancy and the differences are about 7.8% with Forthmann’s experiment and 8.5% with the theoretical value of Kotsovinos.

It was previously argued in the report that the mixing process with the surrounding fluid and the rate of spread are interconnected. The same argument is adopted here in the study of the buoyant jet.

Figure 22 shows that both quantities have the same amount of maximum vortices, but the distance where the maximum vorticity around the z direction is attained is lower above the jet axis than below. This trend is not true further downwind as can be seen in Appendix E; the distance above the jet axis has the maximum vorticity magnitude around the z direction showing the rising phenomenon in the plume dominated region.

With the initial results and discussion from this two-dimensional study with and without buoyancy, it is now possible to advance with confidence to three-dimensional simulations. A more realistic scenario can be adopted; the jet is still far above ground but a co-flowing velocity was added to replicate the headwind velocity.

3 THREE-DIMENSIONAL STUDY OF FREE TURBULENT JET

The first test involves a round jet of diameter 0.93m in an isothermal co-flow. A CFM56-3C engine release with velocity 80m/s was set up, with passive NOx at the exhaust temperature of 690K. The calculation of the assigned mass fraction of NOx was explained in a previous ALAQS report [Ref 21.], and its magnitude is 0.74e-4.

A transient simulation was conducted to monitor the progress of the flow throughout the control volume. The geometry, mesh distribution and boundary conditions are discussed in the following section.

A similar study has only been conducted in a non-buoyant situation. Townsend [Ref 20.] has thoroughly discussed such problem but it was Bradbury [Ref 27.] who derived some of the relationships that will be discussed later in the report.

Both Wygnanski & Fiedler [Ref 7.] and Rodi [Ref 9.] argued that this type of flow is non-self preserving because of the co-flow surrounding it. However, Rodi pointed out that Bradbury’s
results are the most consistent among all experiments in this field because of the high difference in velocity between the jet and its co-flow. This set up results in reducing the “relative intensities” and, as a consequence, minimises the measurement errors [Ref 9.]. Rodi recommended Bradbury’s turbulence data as the most reliable.

3.1 Geometry and Boundary Conditions

The computational domain is of cylindrical shape, composed of five main volumes. This configuration was selected to simplify the problem and optimise the mesh distribution where it is most needed (i.e. near the exhaust and the centre of the jet). The overall geometry is presented in Figure 23; it extends to \(107.5(2h_0)\) in the x, y and z directions, where \(2h_0\) is the jet diameter.

![Figure 23 - Geometry and mesh distribution of the control volume](image)

The boundary conditions are as follows:

- The jet exhaust is set up as a velocity inlet with velocity magnitude 80m/s, releasing hot gases (690K) of which passive NOx is emitted at a mass fraction of 0.74e-4 as described in [Ref 21.].
- The faces adjacent to the exhaust are also defined as velocity inlets, but with a different magnitude of 2.5m/s to replicate the co-flow condition of a headwind.
- The circular boundaries are defined as walls, as they are located far away from the jet.
- The face opposite the jet is the outflow of the control volume.

The method used in this report to solve the general Navier-Stokes equations of fluid flow is Large Eddy Simulation (LES). The basic concept of the LES model is to explicitly solve the larger eddies of the control volume, whereas the smaller eddies are modelled through a filtering process.

3.2 Mean Velocity Profile Comparison

It was previously discussed for the two-dimensional simulation that both buoyant and non-buoyant jets share some degree of similitude near the exhaust but large differences exist further away from it, with the velocity for the buoyant jet decaying much faster than the non-buoyant jet.
This statement is still valid for the three-dimensional case, as shown in Figure 24. The first time step shows slight differences between the two simulations, especially before $\frac{U}{U_m} = 0$. At $t=5s$, the region near the exhaust is quite similar for both simulations but the downwind part is very different. Similar to the 2D study, it appears that the buoyant jet has a rate of centreline decay which is more pronounced than for the non-buoyant jet. In addition, the distance where the velocity ratio is negligible seems to be shorter for the buoyant simulation than for the non-buoyant case. This is again true and much more visible for the next time step.

This sudden decrease of velocity is due to buoyancy coming into play at large distances downwind. In the case of a non-buoyant jet, there is no vertical momentum that can alter the axial velocity, but in a buoyant jet the vertical action of buoyancy lifts the axial velocity component from its centreline axis making it rise. This analysis is corroborated with Figure 25, which shows the rise of the velocity due to buoyancy.

Figure 24 shows a comparison of the mean velocity profile for the simulation of the jet without (left) and with buoyancy (right), at different time steps. To allow an easier comparison, the same scale is used for both figures. It can be seen that, for the non-buoyant jet, the symmetrical pattern is respected through time whereas the buoyant jet shows a break in the symmetry even after 1 second.

At the first time step ($t=1s$), both jets show a very similar pattern closer to the exhaust but the symmetrical pattern is disrupted for the buoyant jet at nearly 4.5m behind it. This behaviour is also shown in Appendix F, where both buoyant and non-buoyant graphs are compared with Tollmien and Goertler analytical solutions, extended to take into account the circular geometry of the jet. In addition to this, the formula developed by Bradbury is also presented, given by [Ref 27.]:

$$\frac{U}{U_m} = \exp\left[-0.6749\eta^2\left(1 + 0.027\eta^4\right)\right]$$

with $\eta = \frac{y}{b}$. This expression was derived experimentally by Bradbury to account for the co-flow situation. Similar to the 2D study of the Gaussian profile, Bradbury’s curve was found to match Goertler’s curve closely near the jet axis, but in the outer part his curve seems to be closer to Tollmien’s.
Figure 25 - Mean velocity profile evolution for non-buoyant and buoyant jets
A comparison between the results for the buoyant and non-buoyant jet simulations shows that the distinguishable potential core region can be recognised at about 2.5m behind the jet. At 5m and from that distance on, it can be seen that both simulations have reached the fully developed region and the differences start to appear, with the buoyant graph showing the point where the jet reaches the maximum velocity is deviated slightly in the positive η direction. The non-buoyant line plot behind the jet, on the other hand, stays close to Tollmien and Goertler analytical results and Bradbury’s relationship. The deviation of the maximum velocity from the centre for the buoyant jet corresponds to the rise of the velocity above the jet axis, caused by buoyancy effects.

Figure 26 depicts the spreading parameter for both cases; however, no experimental results are available for the buoyant jet. The results are plotted for the region above the real axis of the jet in the vertical plane, to highlight the buoyancy effects.

Figure 26 - Angle of spread comparison after 1s above the centre of the jet axis

Figure 26 shows that both curves somewhat follow a linear relationship described by their respective trend line equations but, as discussed earlier, the buoyant jet has a steeper angle of spread above the centreline axis. It is also interesting to see that both plots start with the same value, which corresponds again to the potential core region.

Although this graph was taken up to 16 diameters behind the jet, a plume-like behaviour is not registered after 1 second. As a matter of fact, at this time, only the jet behaviour and the transition between jet-like and plume-like behaviour can be seen. In line with the discussion for the 2D simulations, both buoyant and non-buoyant trend lines have their virtual origins behind the jet exhaust and their values are quite comparable.

Figure 27 shows that, at t=5s, the two simulations depict similar behaviour very close to the exhaust but, as the jet progresses downwind, differences start to appear. Similar to t=1s, the analysis of the vertical velocity profile showed that the point where the maximum axial velocity is reached for the buoyant simulation rises above the centreline axis further behind the jet. This was found to be true at about 5m behind it, the region before being the potential core region as previously discussed. Again, the spreading parameter can be plotted to show the buoyancy effects on the jet.
At this time, it can be seen that the non-buoyant jet still follows the linear pattern represented by Bradbury’s equation. The rate of spread is slightly higher than at the previous time, showing that the jet is still evolving and has not reached its steady-state condition. The buoyant jet, on the other hand, has been subject to noticeable changes especially far downwind where $b/2b_0$ reaches 7 at about 37 diameters.

In addition to the previously found jet and the intermediate regions, a plume region can now be observed further away from the exhaust. As discussed in section 2.3.2 for the 2D study, these three regions are characteristic to buoyant jets. The jet regime has the lowest rate of spread whereas the buoyant far-field regime is characterised by a very high rate of spread. This particular regime starts for this time step at about 22 diameters behind the exhaust and exhibits a very high vertical momentum, distinctive of buoyancy effects.

The results for the third time step (t=10s) are very similar to the previous one. Near the jet exhaust, both buoyant and non-buoyant simulations exhibit the same behaviour, with the region close to about 5m still in the potential jet core. Further behind, the buoyant jet deviates towards the positive $y/b_0$, showing the plume rise above the centreline axis.

Figure 28 shows that the spreading parameter at t=10s resembles that at t=5s. The non-buoyant jet follows a unique trend line. The buoyant jet, on the other hand, can be separated into three trend lines, representing the different regimes that the buoyant jet passes through. The jet region can be compared with the non-buoyant jet simulation in the initial stage of the jet until the turbulence penetration reaches the centreline axis. The rate of spread is quite comparable and both simulations confirm that the virtual origin is situated behind the real position of the exhaust.

Both the intermediate and the plume regions have increased their rate of spreading when compared to Figure 27. The amplification is much more pronounced for the plume region, as it is the portion that most exhibit the buoyancy effects.
Figure 28 - Angle of spread comparison after 10s above the centre of the jet axis

Plume rise results are shown in Figure 29, presenting the evolution of the maximum height of the plume centreline at three time steps (t=5s, 10s and 25s). At t=1s, the flow is still in the pure jet region and does not lift up. The plots show similar profiles from the nozzle exhaust to about 35 diameters behind it. After this distance, differences appear as the flow progresses through time in this transient simulation. At t=25s, the plume has already risen to about 20m from the ground.

Figure 29 Evolution of the maximum height of the plume centreline

3.3 Vorticity Profile Comparison

In order to better understand the role of vortices in the dispersion process of buoyant and non-buoyant jets, it is first interesting to analyse the instantaneous velocity profile taken at each time step. These figures are presented in Appendix G, where an iso-surface of 80m/s is represented with its grid (black part) in addition to the contour plot.

The profile pattern for each simulation, at different time steps, reflects the previous discussion in terms of velocity rise for the buoyant jet. Another interesting point is the presence of local areas where the velocity is lower than the surrounding velocity of 2.5m/s (dark blue). They are almost all located outside the jet and are more numerous near the exhaust.
A close look at the buoyant simulation in Figure 30 shows that the instantaneous velocity of 80m/s is not continuous like in the mean profile, but appears to puff at the exit of the jet exhaust. A similar phenomenon was also reported by Boguslawski et al. [Ref 28.] in their study of hot jets; in addition, they also found the aforementioned local areas where the velocity is lower than the surrounding velocity. Although they did not explain explicitly how this happens and what are the causes of the puffing occurrence, those behaviours can be linked together and correlated with the vorticity parameter.

The same principle of entrained eddies developing at the extremities of the jet, previously discussed in the 2D study and attributed to the jet flow encountering the surrounding irrotational fluid, also applies here. The local areas where the velocity is lower than the surrounding velocity represent the entrainment eddies caused by the jet flow, and it is also these eddies that break the fluid to such puffing pattern.

To confirm such statement, Figure 31 shows that the maximum vorticity magnitude is concentrated near the jet exhaust and, moreover, the distance it is located corresponds approximately to the jet radius above and below the centreline axis. This figure can also be closely matched with plots found in Boguslawski et al.’s report [Ref 28.].
Appendix H shows the evolution of the vortices around the z-axis at different time steps, for both buoyant and non-buoyant jets, together with the instantaneous velocity iso-surfaces of 80m/s. Both simulations can be correlated with the previous 2D comparison; counter-rotating vortices are formed right at the jet exhaust, positive above and negative below the centreline. These are caused by the shearing effects of the jet flow encountering the surrounding fluid moving at a velocity of 2.5m/s. It is interesting to see that the instantaneous velocity iso-surfaces of 80m/s are all situated in between the counter-rotating vortices, and they appear to vanish when coming into contact with each other. This proves Rajaratnam’s comment on how the potential core disappears [Ref 1.], and is also the cause of the contra-rotating vortices penetration to the centreline axis.

The region just after the contact of the counter-rotating vortices is characterised by high intensity and few larger vortices, which further away reduces in intensity and increase in number. As discussed in the 2D section 2.2.3.5, a similar behaviour was found experimentally by Rockwell & Nicolls [Ref 23.] in their study of plane jets. Similar to Browne et al. [Ref 22.], they found that the vortices undergo a “coalescence cascade” which later breaks up and disappears.

Appendix H compared the vortical structures of the buoyant and non-buoyant simulations, and shows that the counter-rotating vortices collide at the centreline faster for the buoyant jet than for the non-buoyant jet. This has a consequence in terms of the potential core length; the length is in fact shorter for the buoyant jet, as can also be seen in Figure 24.

The effects of buoyancy due to vortices can only be observed further downwind. Figure 32 shows a localised portion of the control volume, the scale of which was reduced to better represent the behaviour of the flow. In the buoyant jet simulation, positive rotation around the z-axis occurs at high altitude whereas the non-buoyant simulation shows no vortices at all at that height. The positive rotation around the z-axis is the component that favours the rise in the plume region, confirming the buoyancy effect visible in the velocity profile (Figure 25).

![Figure 32 - Z-vorticity comparison after 5s](image)
4 CONCLUSIONS

The present report aimed initially at validating the CFD code for a free jet condition, by comparison with some established analytical results. The complexity of the simulation was then increased to account for buoyancy effects. The results of both buoyant and non-buoyant simulations were compared to establish their differences.

The 2D and 3D non-buoyant simulations were found to agree with classical results, replicating the characteristic self-preserving behaviour of the jet after the flow development region. The buoyant jet, on the other hand, breaks the symmetrical pattern of the flow showing rise of the axial velocity above the centreline axis.

The length of the potential core was found to vary depending on the turbulence intensity at the exhaust. Its disappearance is a result of turbulence penetration, which leads to the next region (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 increase further along the axis resulting in a decay of the velocity. The non-buoyant jet has a longer potential core because it is not refrained by buoyancy effects.

The spreading rate is linear for the non-buoyant jet and follows very closely the theoretical predictions. The buoyant jet can be split into three different linear regions:

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

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 centreline axis and help the buoyancy jet to rise higher than the non-buoyant jet.

The next step in this research project will be to bring the jet closer to the ground to study the impact of its presence. This will help improving the knowledge of near-field source dynamics, to the benefit of other models such as Gaussian or Lagrangian.
5 REFERENCES


[Ref 10.] FLUENT 6.2 User’s Manual (2003), Fluent Inc, Lebanon, NH, USA.


Appendix A  Turbulence intensity study comparison
Appendix B  Length scale study comparison
Appendix C  Theoretical velocity profile comparison

Appendix D  2D velocity profile comparison of buoyant and non-buoyant jets
Appendix E  2D vortical structure for buoyant jet

Appendix F  Velocity profile comparison after 1 second
ALAQS CFD Comparison of Buoyant and Non-Buoyant Turbulent Jets

Velocity Profile Comparison at 5m behind the jet

Velocity Profile Comparison at 7.5m behind the jet

Velocity Profile Comparison at 10m behind the jet
Appendix G  Instantaneous velocity profile evolution for buoyant and non-buoyant jets

After 1 second

Jet without buoyancy

Jet with buoyancy

After 5 seconds
ALAQS CFD Comparison of Buoyant and Non-Buoyant Turbulent Jets

Jet without buoyancy

Jet with buoyancy

Jet without buoyancy

Jet with buoyancy

After 10 seconds
Appendix H  Vortices around Z-axis for buoyant and non-buoyant jets

After 1 second

Jet without buoyancy

Jet with buoyancy

After 5 seconds

Jet without buoyancy

Jet with buoyancy

After 10 seconds
Appendix I  
Mean temperature contours at different time steps

After 1 seconds

After 5 seconds
Appendix J  Mean NOx concentration contours at different time steps

After 10 seconds

After 1 seconds

After 5 seconds
After 10 seconds
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 BRETWIGNY 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/