AST4-CT-2005-012238

**FAR-Wake**

Fundamental Research on Aircraft Wake Phenomena

Specific Targeted Research Project

Start: 01 February 2005
Duration: 36 months

**LES of turbulent hot jet/vortex interaction**

Prepared by: Laurent Nybelen and T. Marger (CERFACS)
J.F. Boussuge and T. Schönfeld (CERFACS)

---

**Document control data**

<table>
<thead>
<tr>
<th>Deliverable No.</th>
<th>TR 2.1.2 - 5</th>
<th>Due date:</th>
<th>December 2006</th>
</tr>
</thead>
<tbody>
<tr>
<td>Version:</td>
<td>1.0</td>
<td>Task manager:</td>
<td>Jean-Francois Boussuge</td>
</tr>
<tr>
<td>Date delivered:</td>
<td>28 September 2007</td>
<td>Project manager:</td>
<td>T. Leweke</td>
</tr>
<tr>
<td></td>
<td></td>
<td>EC Officer:</td>
<td>J.-L. Marchand</td>
</tr>
</tbody>
</table>

Project co-funded by the European Commission within the Sixth Framework Programme (2002-2006)

**Dissemination Level**

<table>
<thead>
<tr>
<th>PU</th>
<th>Public</th>
<th>x</th>
</tr>
</thead>
<tbody>
<tr>
<td>PP</td>
<td>Restricted to other programme participants (including the Commission Services)</td>
<td></td>
</tr>
<tr>
<td>RE</td>
<td>Restricted to a group specified by the consortium (including the Commission Services)</td>
<td></td>
</tr>
<tr>
<td>CO</td>
<td>Confidential, only for members of the consortium (including the Commission Services)</td>
<td></td>
</tr>
</tbody>
</table>
LES of turbulent hot jet/vortex interaction

L. Nybelen

CERFACS, Centre Européen de Recherche et de Formation Avancée en Calcul Scientifique, 42, Av. G. Coriolis, 31057 Toulouse Cedex, France

Abstract. Temporal Large-Eddy Simulations (LES) of the interaction of a hot jet with a vortex were performed for different flight phases. Two cases are analyzed: the first one, the jet and the vortex are well-separated as in cruise flight. Dynamics is dominated by the jet diffusion and its entrainment around the vortex core. In the second case, corresponding to the high lift configuration, the jet is close enough to the vortex to inject strong axial perturbations in the core, which can lead to a development of instability causing the loss of vortex coherency. Global properties, such as the circulation profile of the vortex and cross-flow kinetic energy are analysed.
# Table of Contents

1 Introduction ................................................................. 4

2 The numerical tool and initial condition ...................... 5
   2.1 The numerical code NTMIX3D ................................. 5
   2.2 Numerical model and initial condition .................... 11

3 Presentation of the numerical results ....................... 15
   3.1 Well-separated configuration as in cruise phase ........ 15
   3.2 Blowing case: the jet close to the vortex ............... 23

4 Conclusion ................................................................. 34
1 Introduction

The interaction between aircraft wake vortices and exhaust jets is of interest to the characterization of the structure of persistent and hazardous trailing vortices during take-off and landing phases and to investigation of the impact of aircraft emissions on the atmospheric environment, in particular in cruise flight phase.

The present work is a part of the Work Package 2 of the FAR-Wake European project on aircraft wakes. This part of the project focuses on a parametrical study of the effect of a hot jet on a single vortex. A background on the matter is provided in the Deliverable D2.0 (Previous work and present knowledge on vortex interactions with jets and wakes [2]). Note this report completes the first CFD study performed in subtask Sbt.2.1.1 on cold jet/vortex interaction (T.R. 2.1.1-3 [3]).

In this context, the goal of the present work is to analyse in detail the dynamics of interaction between an exhaust turbulent hot jet and a trailing vortex for the three different flight phases: approach, take-off and cruise. This document is decomposed into two main sections: first one presented the numerical tool and initial conditions, then the results of simulations.
2 The numerical tool and initial condition

2.1 The numerical code NTMIX3D

The code used for these studies is NTMIX3D, which has been developed by the Centre de Recherche sur la Combustion Turbulente (CRCT) of the Institut Francais du Pétrole (IFP). The parallel code solves the Navier-Stokes equations for a 3D unsteady compressible flow on a regular or irregular Cartesian grid. The solver is devoted to Direct Numerical Simulations and Large-Eddy Simulations.

A nondimensional formulation of the Navier-Stokes equations is used and high accuracy of the solution is guaranteed by a 6th order compact scheme for the discretisation in space, and time integration is performed with a 3rd order Runge-Kutta method. The subgrid-scale model used to take into account the influence of the nonresolved scales on the resolved scales will be the Filtered Structure Function model (Ducros et al. [4]), which proved to provide excellent results for instability studies (Laporte [5], Moet [6]). In particular, the model does not provide any energy dissipation for well-resolved flows corresponding to the stages before the instability saturation and transition to turbulence. The code has been run on the Compaq SC45 and Cray Xd1 computers of CERFACS.

**Governing equations**

The conservation of mass, momentum and energy of a three-dimensional unsteady compressible viscous flow is described by the Navier-Stokes equations. In a 3D Cartesian coordinate system \((x, y, z)\), the Navier-Stokes equations in conservative form can be expressed as follows:

\[
\frac{\partial W}{\partial t} + \frac{\partial}{\partial x}(f - f_v) + \frac{\partial}{\partial y}(g - g_v) + \frac{\partial}{\partial z}(h - h_v) = 0
\]

(1)

The state vector \(W\) is defined as

\[
W = \begin{pmatrix}
\rho \\
\rho u \\
\rho v \\
\rho w \\
\rho E
\end{pmatrix}
\]

(2)

with \(\rho\) the density, \(u, v, w\) are the 3D velocity components, \(p\) the pressure.
and $E$ the total energy ($E = c_v T + 1/2(u^2 + v^2 + w^2)$). The convective fluxes $f$, $g$ and $h$ in the directions $x$, $y$ and $z$ are defined as:

$$f = \begin{pmatrix} \rho u \\ \rho u^2 + p \\ \rho uv \\ \rho vw \\ u(\rho E + p) \end{pmatrix}, \quad g = \begin{pmatrix} \rho v \\ \rho uv \\ \rho v^2 + p \\ \rho vw \\ v(\rho E + p) \end{pmatrix}, \quad h = \begin{pmatrix} \rho w \\ \rho uw \\ \rho w^2 + p \\ \rho vw \\ w(\rho E + p) \end{pmatrix}$$

The viscous fluxes can be expressed as:

$$f_v = \begin{pmatrix} 0 \\ \tau_{xx} \\ \tau_{xy} \\ \tau_{xz} \\ u_j \tau_{xj} - q_x \end{pmatrix}, \quad g_v = \begin{pmatrix} 0 \\ \tau_{yx} \\ \tau_{yy} \\ \tau_{yz} \\ u_j \tau_{yj} - q_y \end{pmatrix}, \quad h_v = \begin{pmatrix} 0 \\ \tau_{zx} \\ \tau_{zy} \\ \tau_{zz} \\ u_j \tau_{zj} - q_z \end{pmatrix}$$

where $q_i$ are the components of the heat flux vector determined by Fourier’s law of heat conduction. Furthermore, $\tau_{ij}$ is the Reynolds stress tensor linked to viscosity as postulated in Stokes’ hypothesis. Sutherland’s viscosity law is used to represent the variation of viscosity with temperature. The pressure is linked to the state vector by the equation of state for a perfect gas.

In NTMIX3D a nondimensional formulation of the Navier-Stokes equations has been used. This formulation is based on reference quantities - $(L_{\text{ref}}, a_{\text{ref}}, \rho_{\text{ref}}, \nu_{\text{ref}})$ in a manner that the nondimensional quantities (denoted by *) are defined as:

$$u_i^* \cdot a_{\text{ref}} = u_i$$
$$t^* \cdot \frac{L_{\text{ref}}}{a_{\text{ref}}} = t$$
$$\rho^* \cdot \rho_{\text{ref}} = \rho$$
$$\nu^* \cdot \nu_{\text{ref}} = \nu$$

where $a_{\text{ref}}$ is usually a sound speed.

**Subgrid-scale model for LES**

The subgrid-scale model used is the Filtered Structure Function model (see Ducros et al. [4] for details). The Navier-Stokes equations are therefore filtered in physical space by a convolution integral with a convolution kernel.
(filter function) characteristic for the filter, which depends directly on the mesh size, and a second filter has been applied namely the Favre’s filter (for compressible flows). Note that the application of the Favre’s filter corresponds to a change in variables, which allows to express the nonlinear terms into the filtered conservative variables.

Subsequently, the Boussinesq hypothesis is used to model the subgrid-stress tensor \( \tau_{ij} = -(\overline{\rho u_i u_j} - \overline{\rho u_i} \overline{\rho u_j}) \), with \( \overline{\rho u} \) the filtered variable by convolution with the filter function, and \( \overline{\rho u} = \rho \overline{u} / \overline{\rho} \) obtained after applying Favre’s filter. This hypothesis defines

\[
\tau_{ij} - \frac{1}{3} \tau_{kk} \delta_{ij} = \mu_t \left( \frac{\partial \overline{\rho u_i}}{\partial x_j} + \frac{\partial \overline{\rho u_j}}{\partial x_i} - \frac{2}{3} \frac{\partial \overline{\rho u_k}}{\partial x_k} \delta_{kk} \right)
\]

(9)

with \( \mu_t = \rho \nu_t \). The subgrid-scale viscosity for the Filtered Structure Function model is written as

\[
\nu_t(x, \Delta_c, t) = 0.00084 \Delta_c \sqrt{F_2^{(3)}(x, \Delta_c, t)}
\]

(10)

where \( \Delta_c = \pi/k_c \) is the cut-off length corresponding to the cut-off wavenumber \( k_c \), which is chosen in spectral space. The exponent of \( F_2 \) indicates the number of times the high-pass filter has been applied to the resolved flow field. The 2\(^\text{nd}\)-order structure function is defined by

\[
F_2(x, r, t) = \langle \| \overline{u}(x + r, t) - \overline{u}(x, t) \|^2 \rangle_{\|r\|=r}
\]

(11)

\( \overline{u} \), \( \overline{x} \) and \( \overline{r} \) denote the velocity vector, position vector and separation vector respectively. \( \langle \rangle \) denotes the statistical average over the entire fluid domain. In practice, the cut-off wavenumber is chosen equal to the cut-off wavenumber imposed by the mesh size \( \Delta_x \): \( k_c = k_x = \pi/\Delta_x \). The cut-off wavenumber is such that the filtered variable \( \overline{u} \) in spectral space is given by

\[
\overline{u} = \begin{cases} 
\hat{u} & \text{if } |k| \leq k_c \\
0 & \text{otherwise}
\end{cases}
\]

(12)

with \( \hat{u} \) the Fourier transform of \( u \). The model is based on the equation governing the evolution of the resolved energy spectrum \( E(k, t) \), dependant on the wavenumber \( k \) and time \( t \):

\[
\left( \frac{\partial}{\partial t} + 2\nu k^2 \right) E(k, t) = T_{<k_c}(k, t) + T_{SG}(k, t)
\]

(13)

with the assumption that \( k < k_c \). \( T_{<k_c}(k, t) \) represents the energy transfer between the resolved scales \( (k < k_c) \) and \( T_{SG}(k, t) \) represents the energy
transfer between the subgrid-scales. This last term requires modeling. The
turbulent viscosity in spectral space is defined by

\[ \nu_t(k, k_c, t) = \frac{T_{SG}(k, t)}{2k^2E(k, t)} \]  

(14)

When this expression is rewritten with respect to the kinetic energy at the
cut-off frequency, one obtains

\[ \nu_t(k/k_c, t) = f(k/k_c) \sqrt{\frac{E(k_c, t)}{k_c}} \]  

(15)

with \( f \) a known function. The expression (15) shows that the model will
not dissipate while there is no energy at the cut-off. This is the case for
the simulations in the present study, where the initial conditions have a
negligible energy level at the cut-off. The energy level at the cut-off increases
through the course of our simulation when locally the flow may transition
and when the turbulent viscosity follows this evolution. This model proves
to be well adapted for our simulations.

**Numerical Methods**

The nondimensionalized Navier-Stokes equations are discretized in space
using a finite difference method. The computational meshes used for the
type of simulations presented in this report are Cartesian, irregular in the
horizontal and vertical transverse directions and regular in the axial direction.

The time integration has been done by a 3\textsuperscript{rd}-order 3-stage Runge-Kutta
method. When one considers \( y \) the solution of Cauchy’s problem \( y' = f(t, y) \),
the integration following Runge-Kutta’s method gives:

\[
\begin{align*}
y(t + \Delta t) &= y(t) + \Delta t \cdot \hat{f}(t, y) \\
\hat{f}(t, y) &= \frac{1}{4} K_1 + \frac{3}{4} K_3 \\
K_1 &= f(t, y) \\
K_2 &= f(t + \Delta t/3, y + \Delta t/3 K_1) \\
K_3 &= f(t + 2\Delta t/3, y + 2\Delta t/3 K_2)
\end{align*}
\]  

(16)

The maximal timestep is given by the limit of stability, expressed usually
as Courant-Friedrichs-Lewy number CFL for the convective part and as the
Fourier number for the diffusive part. These numbers may be expressed by
the dimensionless variables as (in one dimension):

\[
CFL = \text{Max}(u^* + \sqrt{C_p^*.(\gamma - 1).T^*}). \frac{\Delta t^*}{\Delta x^*})
\]

\[
Fo = \text{Max}(\frac{\mu^*}{\rho^*}.\frac{\Delta t^*}{\Delta x^2}.\frac{1}{Re})
\]

(17) (18)

For the case studied \(CFL < 0.5\) and \(Fo < 0.1\) appeared to be sufficient to provide numerical stability. Subsequently, for each iteration the time step is determined by the most strict of either condition over the entire mesh.

For regular grids the spatial derivatives of order 1 (convective terms) and order 2 (diffusive terms) are computed using a 6\(^{th}\) compact scheme [7] (Padé type). For the first order derivatives \((du/dx)(x_i)\) of variable/function \(u\) at a point \(x_i\), the approximation \(u'_i\) is introduced, obtained by a Padé scheme which in it’s general form reads

\[
\beta u'_{i-2} + \alpha u'_{i-1} + u'_i + \alpha u'_{i+1} + \beta u'_{i+2} = \frac{u_{i+3} - u_{i-3}}{6h} + b \frac{u_{i+2} - u_{i-2}}{4h} + a \frac{u_{i+1} - u_{i-1}}{2h}
\]

(19)

In order to obtain a scheme with a truncation error of the order \(O(h^6)\), one has to satisfy the following constraint

\[
a + 2^4b + 3^4c = 2\frac{5!}{4!} (\alpha + 2^4\beta)
\]

(20)

The determination of the first order derivatives reduce to a linear pentadiagonal system with the following form

\[
Au' = Bu
\]

(21)

Solving this system provides the approximation \(u'_i\) in every point. In the particular case with \(\beta = 0\), which is the choice made for NTMIX3D, one obtains a family of schemes that are associated to tridiagonal systems. The other constants of the scheme are \(\alpha = 1/3, a = 14/9, b = 1/9\) and \(c = 0\). Subsequently, the tridiagonal system is inversed by a Thomas algorithm. The prime interest for the use of such schemes is that one obtains high-level accuracy \((O(h^6))\) with a reduced stencil (5 points). Moreover, Lele [7] shows that his schemes possess very weak dispersive errors (quasi-spectral behavior). In particular, the treatment of short-wavelengths by this type of schemes is remarkable. The treatment of boundary conditions brings about the deterioration of the accuracy at the edges of the computational domain
(O(h^3) at the boundary). Upwind compact schemes can be used in the proximity of an edge in the case where nonperiodic boundary conditions are used. For the second order derivatives, a 6th-order tridiagonal scheme is also used.

**Nonuniform mesh**

Nonuniform mesh treatment is required for applications that demand a high resolution of the mesh in a particular region of the computational domain. The use of irregular meshes is a solution in order to avoid an enormous amount of grid points. Several methods exist to perform derivative computations on variable grids such as the Jacobian transformation (JT) method and the method of Fully Included Metrics (FIM), see Gamet *et al.* [8]. The latter method has been implemented in NTMIX3D and has been used to perform the wake vortex simulation. The reason why the FIM method has been retained instead of the JT method is that the JT method can lead to large errors in the case of nonsmoothly varying mesh spacings. In NTMIX3D the FIM method has been applied such that a fourth (respectively third) order approximation is obtained for the first (respectively second) derivative in space in the regions where the grid is stretched.

The FIM method involves direct inclusion of the metrics in the coefficients of the compact derivatives matrices to treat nonuniform meshes. This means adapting the original compact scheme, which is designed for uniform meshes. The main constraint that is imposed is that the obtained scheme for nonuniform meshes reduces exactly to the scheme for uniform meshes in case of a uniformly spaced grid.

In the following only the derivation of the scheme for the first derivative in space is described, as the derivation of the second derivative can be done in a similar manner (for details see Gamet *et al.* [8]). The scheme for the first derivative in space in a general way (similar to Eq. (19)) is given by

\[ \alpha_i f_{i-1} + f_i + \beta_i f_{i+1} = A_i f_{i+1} + B_i f_{i-1} + C_i f_{i+2} + D_i f_{i-2} + E_i f_i \]  

(22)

where the coefficients \( \alpha_i, \beta_i, A_i, B_i, C_i, D_i \) and \( E_i \) are functions of the nonuniform mesh spacings \( \Delta_k = x_k - x_{k-1} \). Following Lele [7], the relation between the former coefficients can be derived by matching the Taylor series coefficients of various orders. The truncation error is determined by the first unmatched coefficient in the Taylor series expansion. The following relations
are obtained:

\[ A_i + B_i + C_i + D_i + E_i = 0 \quad \text{(order 0)} \]

\[ h_{i+1} A_i - h_i B_i + (h_{i+2} + h_{i+1}) C_i - (h_i + h_{i-1}) D_i = 1 + \alpha_i + \beta_i \quad \text{(1st order)} \]

\[ h_{i+1}^2 A_i + h_i^2 B_i + (h_{i+2} + h_{i+1})^2 C_i + (h_i + h_{i-1})^2 D_i = \frac{2}{h_i} (h_{i+1} \beta_i - h_i \alpha_i) \quad \text{(2nd order)} \]

\[ h_{i+1}^3 A_i - h_i^3 B_i + (h_{i+2} + h_{i+1})^3 C_i - (h_i + h_{i-1})^3 D_i = \frac{3}{2h_i} (h_{i+1}^2 \beta_i + h_i^2 \alpha_i) \quad \text{(3rd order)} \]

\[ h_{i+1}^4 A_i + h_i^4 B_i + (h_{i+2} + h_{i+1})^4 C_i + (h_i + h_{i-1})^4 D_i = \frac{4}{3h_i^3} (h_{i+1}^3 \beta_i - h_i^3 \alpha_i) \quad \text{(4th order)} \]

If one wants to obtain a fourth order scheme, the solution can be found in a linear system composed of the five equations given in Eq. (23), where \( A_i, B_i, C_i, D_i \) and \( E_i \) the unknowns. The coefficients are given in a general form in the work of Gamet et al. [8]. In the expressions for the unknowns the parameters \( \alpha_i \) and \( \beta_i \) are considered constants equal to their values for uniform meshes, namely \( \alpha_i = \beta_i = 1/3 \). This implies that the scheme for the first derivative in space exactly reduces to the scheme described previously for uniformly spaced grids. In case the grid does not vary smoothly the accuracy of the scheme reduces to fourth order.

For nonperiodic boundaries, Eq. (22) can not be applied to the points close to the boundary. Therefore, boundary schemes are required at points 1, 2, \( N-1 \) and \( N \). The scheme for the first derivative in space at point 1 is

\[ f'_1 + \alpha f'_2 = A f_1 + B f_2 + C f_3 \quad \text{(24)} \]

This relation can be of third order. At boundary point \( i = 2 \) the scheme for the first derivative in space is given by

\[ \alpha f'_1 + f'_2 + \beta f'_3 = A f_1 + B f_2 + C f_3 \quad \text{(25)} \]

This relation can be of fourth order. The solution coefficients for these two boundary schemes are determined in Gamet et al. [8].

### 2.2 Numerical model and initial condition

The study of hot jet/vortex interaction is identical to the case of cold jet/vortex interaction (see Technical Report TR. 2.1.1-3 [3]). The single
The numerical tool and initial condition
difference between the two studies is the value of jet characteristic parameters (density and velocity).

Two physical assumptions are used for the Large-Eddy Simulation of jet/vortex interaction. The first one is that the temporal evolution of the flow is a good representation of its spatial evolution \([5,9,10]\). This is based on the hypothesis of a locally parallel flow, which means that the gradients of the mean flow in the axial direction are neglected over the short distance corresponding to the axial dimension of the computational domain. The second is that the flow can be split into two phases: the jet regime, where an isolated jet is simulated and the interaction or deflection regime where a vortex is inserted in the turbulent jet flow. This assumption is valid in case of the jet and the vortex are well-separated. This configuration is suitable for cruise flight or for particular high-lift phases of two-engine aircrafts.

The following section describes how the initial values of the characteristics parameters have been determined, and presents the different configurations and details of initial condition (computational domain, mesh, boundaries conditions).

Flow configurations

Two types of interactions are analyzed: in the first one the jet and the vortex are initially well separated. This configuration, called entrainment case, corresponds to a cruise flight condition. The jet regime is first simulated, then the interaction of a turbulent jet and a vortex is resolved. The second type of interaction is called blowing case: the jet blows partially inside the vortex core and is suitable to model high-lift configurations \([10]\).

The extra-deliverable of NLR \([11]\) is used to fix the values of characteristic parameters. In this deliverable are provided the data of a turbofan jet and flight parameters for various operation phases (take-off, cruise and approach idle), allowing an estimation of the vortex circulation. The properties of a representative high by-pass engine double flux jet (turbofan GE CFD6-80C2) were simulated with the NLR Gas turbine Simulation Program GSP for the three flight phases.

However a single jet is used for our study. Thus, an equivalent single hot jet has been determined having the same outflow and thrust. The jet velocity used for the temporal LES and the Reynolds number are summarized in the table 1. The Reynolds numbers \(Re_j = u_j r_j / \nu\) is based on the jet radius, \(r_j / b = 0.03\) \((b: \text{wingspan})\), on exit velocity \(u_j\) and on air viscosity \(\nu\).
The jet axial velocity and density distribution, are defined initially by a tanh profile ($r$ is the radial coordinate in a cross section):

\[
U_{j0}(r) = \frac{1}{2} \left\{ (U_j + U) - (U_j - U)tanh \left[ \frac{1}{4} \frac{r_j}{\theta} \left( \frac{r}{r_j} - \frac{r_j}{r} \right) \right] \right\}
\]

(26)

\[
\rho_{j0}(r) = \frac{1}{2} \left\{ (\rho_j + \rho_a) - (\rho_j - \rho_a)tanh \left[ \frac{1}{4} \frac{r_j}{\theta} \left( \frac{r}{r_j} - \frac{r_j}{r} \right) \right] \right\}
\]

(27)

where $\rho_a$ is the atmosphere density. Note the parameter $S = \rho_j/\rho_a$ of the density ratio is introduced. The co-flow velocity or the airspeed of the aircraft is set to zero $U = 0$ as we perform temporal simulations. A white noise random is superimposed on this velocity component, such as $U_j = U_{j0}(1 + \epsilon \tilde{u})$, where the random perturbation has a maximum amplitude of 0.5% of the local base flow $U_{j0}$, i.e., $\epsilon = 0.005$ and $||\tilde{u}|| \leq 1$. The momentum thickness $\theta$ defined as:

\[
\theta = \int_0^\infty \frac{U(r)}{U_j} \left( 1 - \frac{U(r)}{U_j} \right) dr
\]

(28)

is set equal to $r_j/\theta = 10$ which corresponds to the most unstable case in the theoretical analysis of Michalke and Herman [12]. A passive scalar $Y_t$ is initialized with the same distribution (with $Y_{t_{\text{max}}} = 1$).

The Lamb-Oseen vortex model is used here. The tangential velocity profile is defined as:

\[
v_{\theta}(r, \theta, t) = \frac{\Gamma}{2\pi r} \left( 1 - e^{-\beta (r/r_c)^2} \right)
\]

(29)

where $\beta = 1.2544$. This expression relates the circulation strength $\Gamma$ to the maximum tangential velocity $v_c \equiv v_{\theta}(r_c)$:

\[
\Gamma = 2\pi \epsilon r_c v_c \quad \text{, with } \quad \epsilon = 1.4
\]

(30)

For a given aircraft, the total wake circulation strength can be computed from the classic relation:

\[
\Gamma_0 = \frac{C_l U_b}{2ARs_0}
\]

(31)
where $AR$ is the wing aspect ratio and $s_0$ is the span-wise loading parameter ($s_0 = \pi/4$ for an elliptical loaded wing). The modelling of a vortex interaction with a jet does not realistic in assuming a complete developed vortex with this circulation [11]. Thus, the Lamb-Oseen vortex has a circulation strength of 60% of the total circulation $\Gamma_0$ and its core size is $r_c = 0.5r_j$. The Reynolds number based on the circulation and vortex core size $Re_\Gamma = \Gamma/\nu$ (Tab. 2.2).

The configuration of the jet/vortex interaction can be characterized by the parameter $R$: 

$$R = \frac{\rho_j U_j (U_j - U) A_j}{\rho \Gamma^2}$$

where $U$ is the aircraft velocity (Tab. 2.2) and $A_j = \pi r_j^2$ the jet area.

The separation distance between the vortex center and the jet is defined in the horizontal direction by $\Delta_h$ and vertical direction by $\Delta_v$. All configurations simulated are summarized in Tab. 2.2.

### Table 2. Summary of configurations for the hot jet/vortex interaction. $U$ is the airspeed of the aircraft.

<table>
<thead>
<tr>
<th>Flight Phase</th>
<th>Separation Distance ($\Delta_h/b, \Delta_v/b$)</th>
<th>$U$ [m/s]</th>
<th>$R$</th>
<th>$S$</th>
<th>$Re_j \times 10^6$</th>
<th>$Re_\Gamma \times 10^6$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cruise</td>
<td>(0.145, $-0.058$)</td>
<td>254.8</td>
<td>4.36</td>
<td>0.832</td>
<td>1.74</td>
<td>5.99</td>
</tr>
<tr>
<td>Cruise</td>
<td>(0.29, $-0.058$)</td>
<td>254.8</td>
<td>4.36</td>
<td>0.832</td>
<td>1.74</td>
<td>5.99</td>
</tr>
<tr>
<td>Approach idle</td>
<td>(0, $-0.058$)</td>
<td>68.1</td>
<td>0.94</td>
<td>0.897</td>
<td>3.43</td>
<td>15.55</td>
</tr>
<tr>
<td>Take-off</td>
<td>(0, $-0.058$)</td>
<td>85.1</td>
<td>4</td>
<td>0.885</td>
<td>7.99</td>
<td>15.55</td>
</tr>
</tbody>
</table>

### Numerical frame

The jet regime is simulated in a computational domain defined as $L_x = L_y = 16r_j = 0.465b$ and $L_z = 6r_j = 0.174$ ($r_j$ being the jet radius and $b$ the wingspan). It consists of $N_x \times N_y \times N_z = 161 \times 161 \times 61$ grid points. The length of the axial direction was chosen according to the results of the stability analysis of Michalke and Herman [12], while the transversal lengths were chosen to minimize the effects of boundaries. The mesh is regular for the jet regime with a grid resolution $\Delta = 0.1r_j$ in the three directions.

For the simulation of jet/vortex interaction, the computational domain is larger in the transverse $x, y$–plane, such as $L_x = L_y = 80r_j = 2.323b$ (Fig. 1). In this plane, the mesh is regular in the region of interest for the dynamics of the jet/vortex interaction, i.e. $L_{xp} = L_{yp} = 36r_j = 1.045b$ with
the same grid resolution as in jet regime calculation (Fig. 1). The mesh is then stretched far away to minimize the boundaries effects.

Periodic boundary conditions are used along the jet/vortex axis $z$ and nonreflecting boundary conditions on the $x-y$ directions (treated according to the characteristic approach developed by Poinsot and Lele [13]). Special care has been taken to isolate the interior domain from the nonreflecting boundaries by filtering on the last five grid points every five iterations (see for details [7]).

3 Presentation of the numerical results

3.1 Well-separated configuration as in cruise phase

Jet regime

In this paragraph, the results are presented in non-dimensional form. The velocity is normalized by the airspeed velocity $U$ and the length by the initial jet radius $r_j$. The normalized time is $t_j = t/(r_j/w_j)$, $w_j$ being the initial jet velocity.

The dynamics of a hot jet regime in cruise condition is illustrated in Fig.2 using isosurfaces of density and isocontours of azimuthal vorticity. The jet regime is governed by the development of a Kelvin-Helmholtz instability,
leading to a turbulent jet. This instability is associated to the formation
of structure with high azimuthal vorticity causing the roll-up of the jet,
as shown in Fig. 2. A fully turbulent jet is obtained when the instability
saturates.

The jet instability can be analyzed by the temporal evolution of the most
amplified axial Fourier mode. The growth rate $\sigma_k$ of a given axial Fourier
mode $k$ (associated to the axial wave-number by $k_z = 2\pi k/L_z$) can be de-
termined by $\sigma_k = d\ln(E_k^1/2)/dt$ (see [14]), where $E_k^1/2$ is the square root
of kinetic energy of the corresponding Fourier coefficient. It is obtained by
performing discrete Fourier transforms of square kinetic energy in the axial
direction for each grid point $(x_i, y_j)$, then averaging over the transversal
plane. Figure 3 shows the evolution of the most amplified axial mode which
is slightly lower than the theoretical growth rate of a cold jet ([15]) during
the linear phase of instability. This instability begins to saturate at $t_j \sim 35.$
In the following, one can observed a rapid decay of the jet temperature and
axial velocity (see Fig. 4).

We have chosen the time of jet regime $t_j$ to initialize the interaction regime
when the axial velocity has decreased of 50%, i.e $t_j = 45.6$ or at the down-
stream distance $d/b = 3.6$. The downstream distance $d/b$ is determined
following the relation $d = U \times t$. At this time, the jet is in the "self-similar
period", which is a property of a turbulent jet. Figure 5 shows the mean
axial velocity profiles at different time, and the curves collapse using the
suitable scale $r_{1/2}$ (the distance from the center where the axial velocity is
half of the maximum value). These profiles are obtained by first interpo-
lating $w$ in a polar grid, i.e., $w(x, y, z) \rightarrow w(r, \Theta, z)$, and then averaging
along the axial $z$, and azimuthal $\Theta$ directions (represented by the brackets
< . >). Moreover, the spectra of kinetic energy $E(k)$ in Fig. 6 shows that
the inertial range $-5/3$ slope is well recovered by the simulations for the
larger and mid- length scales.
Fig. 2. Dynamics of hot jet regime in case of cruise condition. Top of bottom: $t_j \sim 13.7/27.4/45.66$. Left side: plots of transverse vorticity $\omega_x$ contours lines, in the $y-z$ plane through the jet centerline [22 levels, solid red lines indicate positive vorticity and dashed blue lines negative vorticity]. Right side: evolution of temperature isocontours and selected isosurface $S = \rho_j/\rho_a = 0.93$. 
3.3 Presentation of the numerical results

Fig. 3. Evolution of the most energetic Fourier coefficient (here, the first one $m = 1$). Its growth rate is $\sigma = 0.024w_j/\Theta$.

Fig. 4. Left side: evolution of the maximum $z$-averaged axial velocity $<w_j>$ and right side of the maximum $z$-averaged temperature. The dot blue lines indicate the value of characteristic parameter and time chosen to initialize the interaction regime.

Fig. 5. Evolutions of nondimensional half-width-scaled velocity profiles $<w_j(r)>$ of the jet, during the "self-similar period" (brackets indicate average in the axial and azimuthal directions).


Fig. 6. Kinetic energy spectra $E(k)$ of the jet flow at $t_j = 45.6$ in cruise condition.

**Entrainment regime in cruise condition:**

The characteristic parameters of aircraft and flight phases (airspeed velocity $U$, lift coefficient $C_l$, aspect ratio $AR$ and wingspan $b$) are used to normalize the results. The vorticity is normalized such as $\omega = \omega b/U$, the velocity $v = v/U$ and length $r = r/b$. The non-dimensional time $\tau = tC_lU/(\pi^4 ARb)$ ([1]) is also introduced and the initial time $\tau = 0$, corresponds to the time when the turbulent jet is introduced in the computational domain with the vortex.

This section presents the results of turbulent hot jet/vortex interaction as it might occur in cruise phase. The numerical initial condition is obtained in adding a single vortex to the turbulent jet in a larger computational domain (see section 2.2). Temporal LES were performed for two separations distances:

- $(\Delta_h, \Delta_v) = (5r_j, -2r_j) = (0.145b, -0.058b)$
- $(\Delta_h, \Delta_v) = (10r_j, -2r_j) = (0.29b, -0.058b)$

These separations distances correspond approximatively to the experimental configurations of ONERA, using the jet radius $r_j$ as reference length scale. The interaction starts when the jet velocity has decreased half its initial value as it is mentioned in previous section. So, the initial velocity ratio between the peaks of axial and tangential velocity is $w_j/v_c \sim 1.47$.

The interaction in case of well-separated configuration is characterized by the entrainment of the turbulent hot jet around the vortex. This dynamics is illustrated in Fig. 7 following the evolution of a selected vorticity isosurface (level $\omega = \omega_{max}/e^\beta$, $\omega_{max}$ being the actual maximum vorticity magnitude) and temperature isocontours. A process of interaction in three stages can be distinguished. Firstly, the jet is entrained around the vor-
3 Presentation of the numerical results

tex due to the velocity induced by the vortex and the pressure gradient. Then, when the jet is enough close to the vortex, the interaction is strong and result in creation of three-dimensional structures of azimuthal vorticity around the vortex. The vortex shape is also disturbed in longitudinal direction, while the temperature contained in the jet continues with decrease towards the ambient temperature. The last stage is characterized by the decay of these structures and the vortex finds again a cylindrical shape.

The evolution of the maximum azimuthal vorticity \( \omega_{\theta_c}^{\text{peak}} \) (Fig. 8) in a domain of \( 2r_c \) \( (r_c \text{ being the vortex core radius}) \) surrounding the core allows to display the times of the three stages of interaction regime (see [9]). One can observe that the separation distance between the jet and the vortex influences on the interaction times and on the level of maximum azimuthal vorticity around the vortex. The interaction process is more pronounced when the jet and the vortex are close. The comparison with the turbulent cold jet/vortex interaction (identical vortex, see for details [3]) shows that the interaction takes place earlier. This can be explained by the fact that the hot jet is lightweighter, even if the temperature decreases rapidly during the interaction (Fig. 9).

The cross flow kinetic energy \( E_k \) is defined as

\[
E_k = \frac{1}{2} \rho \int (u^2 + v^2) dS
\]

(33)

The integration area corresponds to the regular domain of transverse \( xy \) plane. The velocity field is first averaged in axial direction before to calculate the cross flow kinetic energy. This latter starts to decrease when the interaction is strong (Fig. 10), corresponding to the time when the maximum azimuthal vorticity is reached. In both cases, the evolution of cross-flow kinetic energy has the same behaviour. For the longer distance \( ((\Delta h, \Delta v) = (0.29b, -0.058b)) \) it is similar to the cold configuration, while for the other case, we can observe that it starts to decrease earlier and attains a slower value at the end of simulation \( (\tau \sim 0.225) \).

To determine the evolution of vortex circulation and tangential velocity profile the Cartesian velocity field is first averaged in axial direction, then interpolated in a polar grid following a 3\(^{rd}\) Lagrangian method and finally averaged in azimuthal direction. The vorticity field is integrated on this polar grid, which provides the circulation profile as a function of the radius
3 Presentation of the numerical results

Fig. 7. Dynamics of interaction regime in case of cruise condition and \((\Delta k, \Delta e) = (0.145b, -0.058b)\). Left side: evolution of a selected isosurface \(\omega = \omega_{\text{max}}/e^\beta\) and isocontours of vorticity magnitude. Right side: detailed view of temperature field averaged in axial direction.
taken from the vortex center

$$\Gamma(r) = \int \omega(r, \theta) r dr d\theta$$  \hspace{1cm} (34)

The tangential velocity profile is then determined following the definition

$$v_\theta(r) = \frac{\Gamma(r)}{2\pi r}$$  \hspace{1cm} (35)

The vortex center is localized by the maximum of axial vorticity in the averaged transverse plane. Note, although the tangential velocity profile is well defined by Eq. 35, it has only a physical meaning for the case of an axisymmetric vortex. For this reason, these velocity profiles have an approximate significance as the vortex is deformed in radial and axial directions.

The circulation $\Gamma(r)$ profiles in Fig. 11 show the propagation of an overshoot without attain the inner region of the vortex. The maximum of tangential velocity decreases stronger for the shorter separation distance. The vortex core is affected by the jet to the same manner for the two separation distances. But the impact of the propagation overshoot on the vortex profiles is more pronounced if the distance is short. In both case, the vortex core size increases slightly (see Fig. 11). These observations on the vortex represents a significant difference with the take-off or approach case where the jet and the vortex are close, that is analysed in the next section.

![Fig. 8.](image)

**Fig. 8.** Evolution of the maximum azimuthal vorticity, $\omega_{\theta_{\text{peak}}}$, around the vortex core, during the jet/vortex interaction phase in cruise configuration. Left: case ($\Delta_h, \Delta_v) = (0.145b, -0.058b)$ and, right: ($\Delta_h, \Delta_v) = (0.29b, -0.058b)$ case. Solid red line: hot jet and dashed blue line: cold jet.
3 Presentation of the numerical results

![Graph](image)

Fig. 9. Evolution of the temperature maximum during the jet/vortex interaction phase in cruise configuration. Solid red line: for the initial separation distance $(\Delta_h, \Delta_v) = (0.145b, -0.058b)$ and dashed line: for $(\Delta_h, \Delta_v) = (0.29b, -0.058b)$.

![Graphs](image)

Fig. 10. Evolution of the cross-flow kinetic energy $E_k$ during the jet/vortex interaction phase in cruise configuration. Left side: case $(\Delta_h, \Delta_v) = (0.145b, -0.058b)$ and, right side: $(\Delta_h, \Delta_v) = (0.29b, -0.058b)$ case. Solid red line: hot jet and dashed blue line: cold jet.

3.2 Blowing case: the jet close to the vortex.

The configuration where the hot jet is close to the vortex is analysed in this section. It corresponds to the interaction of the jet and flap vortex during approach/take-off phases. This configuration is called blowing case since the jet blows partially inside the vortex. The jet is initially located just below the vortex such as $(\Delta_h, \Delta_v) = (0.145b, -0.058b) = (0.29b, -0.058b)$. The initial value of velocities ratio between the jet velocity $w_j$ and maximum of tangential velocity $v_c$ of vortex, are: $w_j/v_c \sim 1.816$ in approach condition and $w_j/v_c \sim 4.29$ in take-off condition. Contrary to the well-separated configuration as in cruise phase, the interaction starts immediately. So the assumption to split the simulation in two steps, a jet regime then an inter-
action regime, does not satisfactory. For this reason the jet flow considered is laminar at the beginning of the interaction.

The dynamics is illustrated by the evolution of a selected isosurface of vorticity magnitude and reported in Fig. 12 for the approach case and in Fig. 13 for the take-off case. Both cases are characterized by a rapid roll-up of the jet around the vortex and by a creation of three-dimensional structures of azimuthal vorticity surrounding the vortex. However, in case of take-off configuration the vortex loss its coherence while in case of approach condition it is well disturbed. In the following, the three-dimensional vorticity structures decay due to the complex interactions of small co- and counter-rotating structures.
This interaction process makes this flow similar to a swirling jet or a Batchelor vortex (\(q\)-vortex [17]). The loss of vortex structure in case of take-off condition, could be explained by the development of an instability as in a \(q\)-vortex when the swirl parameter is low [18]. The swirl parameter relates peak axial \((w_j)\) and tangential \((v_c)\) velocities

\[
q = \sqrt{\beta \alpha \frac{v_c}{w_j}} \sim 1.57 \frac{v_c}{w_j}
\]

Its initial value is \(q \sim 0.86\) in case of the approach condition and \(q \sim 0.36\) in case of the take-off condition. These values are below to the stability limit of \(q\)-vortices \(0 < q < 1.5\) [18]. However, it seems that this limit is different in our cases. This can be explained by the difference with a \(q\)- or Batchelor vortex where all axial flow is concentrated in the core; here, axial momentum rapidly decays, leaving the vortex core almost unaffected, as show by the evolution of axial velocity isocontours in Fig. 14,15. Note that the initial vortex position is slightly displaced during the interaction.

As in cruise configuration, the interaction results in a generation of azimuthal vorticity as shows the evolution of maximum azimuthal vorticity in Fig. 18, evaluated around the vortex core \(\omega_{\text{core}}^{\text{peak}}\) and in the transverse plane \(\omega_{\text{core}}^{\text{peak}}\). One can observed the interaction process in three stages already observed in cruise configuration. Moreover it is more significant in case of take-off condition. In both cases, the evolution of maximum temperature in Fig. 19 is characterized by a decay when the jet becomes turbulent \((\tau \sim 0.018\) in take-off condition and \(\tau \sim 0.038\) in approach condition) and tends rapidly towards the ambient temperature. As for the axial jet velocity in case of approach condition, one can observe in Fig. 16 that the temperature does not penetrate into the vortex core. It is equally observed in take-off configuration (Fig. 17) until the moment where the vortex loss its coherence. Due to the turbulence, the cross-flow kinetic energy decreases (Fig. 19). At the end of simulations it reaches a reduction of \(\sim 18\%\) for approach condition and \(\sim 34\%\) for take-off configuration.

The circulation and velocity profiles of the vortex are reported in Fig. 20. The jet affects the vortex by reducing its peak of velocity. In case of approach condition, the vortex does not completely annihilated by the jet as show the tangential velocity profile. In both cases, the total circulation remains constant far from the vortex core.
3 Presentation of the numerical results

Fig. 12. Dynamics of interaction regime in case of approach condition. Evolution of a selected isosurface $\omega = \omega_{\text{max}}/e^\beta$ and isocontours of vorticity magnitude (cut planes).
3 Presentation of the numerical results

Fig. 13. Dynamics of interaction regime in case of take-off condition. Evolution of a selected isosurface $\omega = \omega_{\text{max}}/e^\beta$ and isocontours of vorticity magnitude (cut planes). Note that the selected isosurface is the same on the figures c) and d).
Fig. 14. Evolution of axial velocity isocontours in case of approach condition. Initial vortex position is illustrated by the black circle (respect vortex core radius).
3 Presentation of the numerical results

![Graphs showing axial velocity isocontours with different values of $\tau$.]

a) $\tau \sim 0.0067$

b) $\tau \sim 0.0181$

c) $\tau \sim 0.04$

d) $\tau \sim 0.0608$

**Fig. 15.** Evolution of axial velocity isocontours in case of take-off condition. Initial vortex position is illustrated by the black circle (respect vortex core radius)
3 Presentation of the numerical results

Fig. 16. Evolution of temperature isocontours in case of approach condition. Initial vortex position is illustrated by the black circle (respect vortex core radius).
3 Presentation of the numerical results

Fig. 17. Evolution of temperature isocontours in case of take-off condition. Initial vortex position is illustrated by the black circle (respects vortex core radius)
Fig. 18. Evolution of the maximum azimuthal vorticity, $\omega_{\text{peak}}$, around the vortex core, during the jet/vortex interaction phase in cruise configuration. Solid line with triangle symbols for the maximum of azimuthal vorticity in global transverse plane and dashed line with circle symbols for the maximum of azimuthal vorticity around the vortex core ($r < 2r_c$).

Fig. 19. Left side: evolution of the temperature maximum. Right side: evolution of the cross-flow kinetic energy $E_k$. Dashed line with circle symbols for take-off configuration and solid line with triangle symbols for approach configuration.
Fig. 20. Turbulent hot jet/vortex interaction: circulation $\Gamma(r)$ and tangential velocity $v_{\theta}(r)$ profiles as a function of radial distance from the vortex center, during the jet/vortex interaction phase. Left side: approach condition and right side: take-off phase. The initial jet position is indicated by a point on the $r$-coordinate.
4 Conclusion

Three dimensional temporal Large-Eddy simulations were carried out to study the interaction between an exhaust turbulent hot jet and a vortex during the different flight phases, approach, take-off and cruise. The simulations were performed in two steps. It consists in first simulating the jet regime allowing to obtain a turbulent hot jet, then its interaction with the wake vortex.

Two types of interactions were analysed: in the first case, the jet and the vortex are initially well separated modelling an interaction in cruise condition between the wing tip vortex and the jet. The dynamics of interaction is mainly controlled by the entrainment of the jet by the vortex and the turbulent diffusion of the jet. Finally the solid-body rotation of the vortex core prevents hot temperature to penetrate inside the vortex.

In the second case they are close corresponding to the approach and take-off phases of a four-engine aircrafts (interaction between the external jet and flap vortex). The strong injection of axial flow perturbations lead to the lost of vortex coherence in case of take-off condition, contrary to the approach configuration, where the axial perturbation is weaker. For both cases considered here, the density has not effect on the interaction process but only on the development of the jet. Moreover, the jet effect is to reduce the peak of tangential velocity of the vortex ($30 – 60\%$).

These results, similar to the cold jet/vortex interaction (T.R [3]), revealed that the vortex is really affected by the jet when they are close and when the intensity of axial perturbation is high.
4 Conclusion

References