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 cold jet/vortex interaction

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

Document control data

<table>
<thead>
<tr>
<th>Deliverable No.</th>
<th>Due date</th>
<th>Version</th>
<th>Task manager</th>
</tr>
</thead>
<tbody>
<tr>
<td>TR 2.1.1 - 3</td>
<td>December 2006</td>
<td>1.0</td>
<td>Jean-Francois Boussuge</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Date delivered</th>
<th>Project manager</th>
<th>EC Officer</th>
</tr>
</thead>
<tbody>
<tr>
<td>22 December 2006</td>
<td>T. Leweke</td>
<td>J.-L. Marchand</td>
</tr>
</tbody>
</table>

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

<table>
<thead>
<tr>
<th>Dissemination Level</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>PU</td>
<td>Public</td>
</tr>
<tr>
<td>PP</td>
<td>Restricted to other programme participants (including the Commission Services)</td>
</tr>
<tr>
<td>RE</td>
<td>Restricted to a group specified by the consortium (including the Commission Services)</td>
</tr>
<tr>
<td>CO</td>
<td>Confidential, only for members of the consortium (including the Commission Services)</td>
</tr>
</tbody>
</table>
LES of turbulent cold 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 turbulent cold 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 approach or take-off aircraft phase, the jet is close enough to the vortex to inject a strong axial perturbations in the core, which can lead to a development of instabilities causing the loss of vortex coherence. Global dynamics and vortex properties, such as the circulation profile of the vortex and cross-flow kinetic energy, are analysed.
# Table of Contents

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

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

3. Presentation of the numerical results ............................ 15
   3.1 Well-separated configuration .................................. 15
   3.2 Blowing case: the jet close to the vortex. ................. 25

4. Conclusion .......................................................... 31
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.

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 cold 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]).

In this context, the goal of the present work is to analyze in detail the dynamics of interaction between an exhaust turbulent cold 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 Numerical tool and initial condition

2.1 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. [3]), which proved to provide excellent results for instability studies (Laporte [4], Moet [5]). 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

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

The state vector \(W\) is defined as

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

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:

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

(3)

The viscous fluxes can be expressed as:

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

(4)

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

$$u_i^* \cdot a_{\text{ref}} = u_i$$

(5)

$$t^* \cdot \frac{L_{\text{ref}}}{a_{\text{ref}}} = t$$

(6)

$$\rho^* \cdot \rho_{\text{ref}} = \rho$$

(7)

$$\nu^* \cdot \nu_{\text{ref}} = \nu$$

(8)

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

**Subgrid-scale model for LES**

The subgrid-scale model used is the Filtered Structure Function model (see Ducros et al. [3] 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 \( \tau_{ij} = -(\overline{\rho u_i u_j} - \overline{\rho \tilde{u}_i \tilde{u}_j}) \), with \( \overline{\rho} \) the filtered variable by convolution with the filter function, and \( \tilde{u} = \overline{\rho u} / \overline{\rho} \) obtained after applying Favre’s filter. This hypothesis defines

\[
\tau_{ij} - \frac{1}{3} \tau_{kk} \delta_{ij} = \mu_t \left( \frac{\partial \tilde{u}_i}{\partial x_j} + \frac{\partial \tilde{u}_j}{\partial x_i} - \frac{2}{3} \frac{\partial \tilde{u}_k}{\partial x_k} \delta_{ij} \right)
\]

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

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

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\(^{nd}\)-order structure function is defined by

\[
F_2(\overline{x}, r, t) = \langle \| \overline{u}(\overline{x} + \overline{x}', t) - \overline{u}(\overline{x}', t) \|_T^2 \rangle_{\| \overline{x}' \|_T}
\]

\( \overline{u}, \overline{x} \) and \( \overline{x}' \) 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 \( \tilde{u} \) in spectral space is given by

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

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 \( t \)

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

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

$$\nu_t(k, k_c, \tau) = \frac{T_{SG}(k, \tau)}{2k^2E(k, \tau)}$$

(14)

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

$$\nu_t(k/k_c, \tau) = f(k/k_c)\sqrt{\frac{E(k_c, \tau)}{k_c}}$$

(15)

with $f$ a known function. Eq. (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 3rd-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

$$y(t + \Delta t) = y(t) + \Delta t \cdot \hat{f}(t, y)$$

$$\hat{f}(t, y) = 1/4 \ K_1 + 3/4 \ K_3$$

$$K_1 = f(t, y)$$

$$K_2 = f(t + \Delta t/3, y + \Delta t/3 \ K_1)$$

$$K_3 = f(t + 2\Delta t/3, y + 2\Delta t/3 \ K_2)$$

(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)

\[
CFL = \text{Max}((u^* + \sqrt{C_p^* (\gamma - 1) T^*}) \frac{\Delta t^*}{\Delta x^*}) \quad (17)
\]

\[
Fo = \text{Max}(\frac{\mu^*}{\rho^*} \frac{\Delta t^*}{\Delta x^2} \frac{1}{Re}) \quad (18)
\]

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 6th order compact scheme [6] (Pade 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 Pade scheme which in it’s general form reads

\[
\beta u_{i-2} + \alpha u_{i-1} + u_i' + \alpha u_{i+1} + \beta u_{i+2} =
\frac{c}{6h} (u_{i+3} - u_{i-3}) + \frac{b}{4h} (u_{i+2} - u_{i-2}) + \frac{a}{2h} (u_{i+1} - u_{i-1}) \quad (19)
\]

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

\[
a + 2^4b + 3^4c = 2^5 \frac{5!}{4!} (\alpha + 2^4\beta) \quad (20)
\]

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

\[
Au' = Bu \quad (21)
\]

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 \(\alpha = 1/3, \quad a = 14/9, \quad b = 1/9\) 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 [6] shows that his schemes possesses 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 vicinity 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. [7]. 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 non-smoothly 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. [7]).

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 [6], 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

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

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

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

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

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

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. [7]. 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

\[
f'_1 + \alpha f'_2 = Af_1 + Bf_2 + Cf_3 \quad \text{(24)}
\]

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

\[
\alpha f'_1 + f'_2 + \beta f'_3 = Af_1 + Bf_2 + Cf_3 \quad \text{(25)}
\]

This relation can be of fourth order. The solution coefficients for these two boundary schemes are determined in Gamet et al. [7].

### 2.2 Numerical model and initial condition

This present study of cold jet/vortex interaction is based on two physical assumptions. The first one is that the temporal evolution of the flow is
a good representation of its spatial evolution [4,8,9]. 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 assumption is to assume that the flow can be split into two phases for the well-separated configuration (cruise phase): 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.

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, is suitable for cruise flight or for particular high-lift phases of two-engine aircrafts. The jet regime is first simulated, then the interaction of a turbulent jet and a vortex is resolved. The second type of interaction is characterized by a short separation distance between the jet and the vortex, and is suitable to model high-lift configurations [9].

The extra-deliverable of NLR [10] is used to fix the values of characteristic parameters. In this deliverable the data are provided 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 (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 in a first time having the same outflow and thrust. Then, the cold jet velocity is deduced to this data of single hot jet. The jet velocity used for the temporal LES and the Reynolds numbers are summarized in the table 1. The Reynolds numbers $Re_j = w_j r_j / \nu$ is based on the jet radius, $r_j/b = 0.03$ $(b$: wingspan), on exit velocity $w_j$, and on air viscosity $\nu$.

The jet axial velocity is defined by a tanh profile ($r$ is the radial coordinate in a cross section)

$$ w_{j0}(r) = \frac{1}{2} \left\{ (w_j + U) - (w_j - U) \tanh \left[\frac{1}{4} \frac{r_j}{\theta} \left( \frac{r}{r_j} - \frac{r_j}{r} \right) \right] \right\} \quad (26) $$
2 Numerical tool and initial condition

<table>
<thead>
<tr>
<th></th>
<th>Approach idle</th>
<th>Take-off</th>
<th>Cruise at 8240m</th>
</tr>
</thead>
<tbody>
<tr>
<td>$Re_j$</td>
<td>$5.53 \times 10^6$</td>
<td>$1.38 \times 10^6$</td>
<td>$1.59 \times 10^6$</td>
</tr>
<tr>
<td>$M = w_j/c$</td>
<td>$0.1862$</td>
<td>$0.4657$</td>
<td>$0.1212$</td>
</tr>
</tbody>
</table>

Table 1. The Reynolds and Mach number of single jet for the different flight phases (c sound velocity) used for the temporal LES.

A random white noise is superimposed on this velocity component, such as $w_j = w_{j0}(1 + \epsilon \tilde{w})$, where the random perturbation has a maximum amplitude of 0.5% of the local base flow $w_{j0}$; i.e., $\epsilon = 0.005$ and $||\tilde{w}|| \leq 1$. The momentum thickness $\theta$ defined as

$$\theta = \int_0^\infty \frac{w(r)}{w_j} \left(1 - \frac{w(r)}{w_j}\right) dr$$

is set equal to $r_j/\theta = 10$ which corresponds to the most unstable case in the theoretical analysis of Michalke and Herman [11]. 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

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

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

$$\Gamma = 2\pi \epsilon r_c v_c \quad \text{with} \quad \epsilon = 1.4$$

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

$$\Gamma_0 = \frac{C_l U_b}{2s_0 AR}$$

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 is not realistic in assuming a complete developed vortex with this circulation [10]. DLR proposed to use a circulation strength of 50% of the total circulation $\Gamma_0$ ([10]) and an initial core size in the order of 1% of the wing span. However, we have chosen a circulation strength of 60% of $\Gamma_0$ and an initial core size $r_c/b = 0.015$ ($r_c = 0.5r_j$) to have an acceptable numerical resolution. The Reynolds number based on the circulation $Re_\Gamma = \Gamma/\nu$ is summarized in Tab. 2.2.
The configuration of the jet/vortex interaction can be characterized by the parameter $R$

$$R = \frac{\rho_j w_j (w_j - U) A_j}{\rho I^2} \quad (31)$$

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>
<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>$Re_j \times 10^6$</th>
<th>$Re_F \times 10^6$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cruise</td>
<td>$(0.145, -0.058)$</td>
<td>254.8 1.747 1.59 5.99</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Cruise</td>
<td>$(0.29, -0.058)$</td>
<td>254.8 1.747 1.59 5.99</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Approach idle</td>
<td>$(0, -0.058)$</td>
<td>68.1 0.825 5.33 15.55</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Take-off</td>
<td>$(0, -0.058)$</td>
<td>85.1 3.824 13.83 15.55</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Table 2. Summary of configurations for the cold jet/vortex interaction. $U$ is the aircraft velocity.

**Numerical frame**

The jet regime is simulated in a computational domain defined as $L_x = L_y = 16 r_j = 0.465 b$ and $L_z = 6 r_j = 0.174 b$ ($r_j$ 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 [11], while the transversal lengths were chosen to minimize the effects of boundaries. The mesh is regular for the jet regime with a grid resolution $\Delta = 0.1 r_j$ in the three directions.

For the simulation of jet/vortex interaction in cruise configuration, the computational domain is larger in the transverse $xy$–plane, such as $L_x = L_y = 80 r_j = 2.323 b$ (Fig. 1). In this plane, the mesh is regular in the region of interest for the dynamics of the jet/vortex interaction, i.e. $L_{xp} = L_{yp} = 18 r_j = 0.523 b$ 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 $x$ and $y$ directions (treated according to the characteristic approach developed by Poinset and Lele [12]). Special care has been taken to isolate the interior domain from the nonreflecting boundaries by inserting a buffer zone (five grid points) where the
turbulent viscosity was artificially increased with a linear law.

3 Presentation of the numerical results

3.1 Well-separated configuration

This section presents the jet/vortex interaction as in a cruise configuration, where the jet and vortex are well-separated. The simulations have been 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.

Jet regime

In this paragraph, the results are presented in non-dimensional form. The velocity is normalized by the aircraft 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 jet regime in a cruise condition is presented in this section. It is illustrated in Fig. 2 using isosurfaces of passive scalar and isocontours of azimuthal vorticity in the plane through the jet centerline. The jet transition to the turbulence through a Kelvin-Helmholtz instability mechanism is well represented. 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 determined by $\sigma_k = d\ln(E_k^2/2)/dt$ (see [13]), 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 a good agreement with the theoretical prediction of the growth rate $\sigma$ [14] of the most amplified mode during the jet regime.

The temporal evolutions of the maximum axial velocity and passive scalar are reported in Fig. 4, which shows the rapid decay from $t_j \sim 37$ corresponding to the beginning of transitional phase (saturation of jet instability). These quantities are extracted from the average velocity field in axial direction (denoted by $< >$).

We have chosen the time of jet regime $t_j$ to initialize the interaction regime for the cruise condition when the axial velocity has decreased of 50%, i.e., $t_j = 44.84$ or at the downstream distance $d/b = 8.92$. The downstream distance $d/b$ is determined following the relation $d = U \times t$ (Tab. 2.2). At this time, the jet is in the “self-similar period” as show by the mean axial velocity profile in Fig. 5, where 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 interpolating $w$ 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 simulation for the larger and mid-length scales.
Fig. 2. Dynamics of jet regime in case of cruise condition. Top of bottom: $t_j = 18.2/29.1/44.8$. Left side: plots of transverse vorticity $\omega_z$ contours lines, in the $y - z$ plane through the jet centerline [32 levels, solid/dashed lines indicate positive/negative vorticity]. Right side: evolution of a selected passive scalar isosurface, $Y_t = 0.4$. 
3 Presentation of the numerical results

**Fig. 3.** Evolution of the most energetic Fourier coefficient (the first one in the present cases $m = 1$). Dashed line represents the theoretical growth rate $\sigma = 0.026w_j/\Theta$.

**Fig. 4.** Evolutions of the maximum $z$-averaged axial velocity $< w_j >$ and maximum $z$-averaged passive scalar $< Y_t >$ during the jet regime.

**Fig. 5.** Evolution of non-dimensional half-width-scaled velocity profiles $< w(r) >$, during the "self-similar period" (brackets indicate average in the axial and azimuthal directions).
Presentation of the numerical results

Fig. 6. Kinetic energy spectra $E(k)$ of the jet flow at $t_j = 44.84$ for the cruise condition.

**Entrainment regime in a cruise condition:**

The characteristic parameters of aircraft and flight phases (speed of the aircraft $U$, 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 $\tau = t \times 16C_l U/(\pi^4 ARb)$ ([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 cold jet/vortex interaction as they 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 corresponds 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.

The dynamics of the interaction in case of the jet and the vortex are well-separated, is dominated by the entrainment of the jet inside the vortex field. This process is illustrated in Fig. 7 which displays the evolution of a selected vorticity isosurface (level $\omega = \omega_{max}/e^{\beta}$, $\omega_{max}$ being the actual maximum vorticity magnitude) and passive scalar isocontours. Three stages can be distinguished. Firstly, the vortex velocity induces the entrainment of the jet around the core. Then, the jet is enough close to the core, to
3 Presentation of the numerical results

strongly interact with the vortex tangential velocity, causing the creation of three-dimensionnal structures of azimuthal vorticity around it. At the end of interaction these structures progressively decay and only the vortex core remains visible. Note that the jet remains outside of the inner region as shown by the isocontours of passive scalar (initially contained in the jet field) averaged in axial direction. The evolution of the maximum azimuthal vorticity $\omega_{\text{peak}}^c$ (Fig. 8) in a domain of $2r_c$ ($r_c$ being the vortex core radius) surrounding the core allows to display the times of the three stages of interaction regime (see [8]). 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.

Figure 9 shows the $\omega_x$ contours field on a plane passing through the vortex center. The axial vorticity isocontour $\omega_z = \omega_{z_{\text{max}}} / e^\beta$ is superimposed to visualize the vortex core. Increasing transverse vorticity structures of opposite sign (contrarotative) are generated around the vortex during the interaction regime. Their mutual interactions lead successively to another creation and decay.

The cross flow kinetic energy $E_k$ is defined as

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

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 the two cases of separation distance, the cross flow kinetic energy has decreased at the end of interaction regime of 20% of its initial value.

To determine the evolution of the vortex characteristics during the interaction regime, the Cartesian velocity field is first averaged in axial direction, then interpolated in a polar grid following a Lagrangian method of 3rd order 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 taken from the vortex center

$$\Gamma(r) = \int \omega(r, \theta) rdrd\theta$$

The tangential velocity profile is then determined following the definition

$$v_\theta(r) = \frac{\Gamma(r)}{2\pi r}$$
Fig. 7. Dynamics of interaction regime in case of cruise condition and \((\Delta h, \Delta v) = (0.145b, -0.058b)\). Left side: evolution of a selected isosurface and isocontours of vorticity magnitude \(\omega = \omega_{max}/e^3\). Right side: detailed view of passive scalar field averaged in axial direction.
3 Presentation of the numerical results

![Graph showing the evolution of the maximum azimuthal vorticity, \( \omega_{\theta_{\text{peak}}} \), around the vortex core, during the jet/vortex interaction phase in cruise configuration. Solid black line: \((\Delta_h, \Delta_v) = (0.145b, -0.058b)\) and dashed blue line \((\Delta_h, \Delta_v) = (0.29b, -0.058b)\).

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. 34, 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 averaged circulation \( \Gamma_{5-15} \) is defined as

\[
\Gamma_{5-15} = \frac{\int_{5}^{15} \Gamma(r) dr}{\int_{5}^{15} dr}
\]

This corresponds to an integration between \( r/b = 0.114 \) and \( r/b = 0.342 \). This circulation increases of maximum 2%. The maximum of tangential velocity decreases of 30% while the vortex core size increases approximately of 10% during the interaction regime (see Fig. 11). The vortex core is affected by the jet to the same manner for the two separation distances. 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.
3 Presentation of the numerical results

Fig. 9. Case of cruise condition and \((\Delta_h, \Delta_z) = (0.145b, -0.058b)\). Evolution of azimuthal vorticity contours lines (40 levels, \(\omega_x \in [\omega_{x_{\min}}, \omega_{x_{\max}}]\)) in a transversal plane through the vortex center, left to right: at \(\tau \sim 0.056/0.106/0.283\). The axial isocontour \(w_z = w_{z_{\max}}/e^{\beta}\) is superimposed (black solid line).
3 Presentation of the numerical results

Fig. 10. Evolution of the cross-flow kinetic energy $E_k$ during the jet/vortex interaction phase in cruise configuration. Solid black line: $(\Delta h, \Delta v) = (0.145b, -0.058b)$ and dashed blue line $(\Delta h, \Delta v) = (0.29b, -0.058b)$.

Fig. 11. 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 (cruise configuration). Left side: $(\Delta h, \Delta v) = (0.145b, -0.058b)$ and right side: $(\Delta h, \Delta v) = (0.29b, -0.058b)$. The initial jet position is indicated by a point on the r-coordinate.
3.2 Blowing case: the jet close to the vortex.

The configuration where the jet is close to the vortex is analysed in this section, corresponding to the interaction of the jet and flap vortex during approach and take-off phases. This interaction can be called blowing case since the jet blows partially inside the vortex. The jet is initially located just below the vortex such as $\Delta_h, \Delta_v = (0.7, -2.7) = (0.6, -0.058b)$. The jet is laminar for this configuration as the separation distance is too low to assume two regimes (jet and interaction regime as in well-separated configuration). The initial value of velocities ratio between the jet velocity $w_j$ and maximum of tangential velocity $v_c$ of vortex, are: $w_j/v_c \sim 1.55$ in approach condition and $w_j/v_c \sim 3.88$ in take-off condition.

The three dimensional dynamics is illustrated by a selected isosurface of vorticity magnitude and reported in Fig. 12. In these configurations, the jet rapidly wraps around the vortex leading to a strong deformation of the vortex structure which finally loss its coherence. This observation is more visible in case of take-off condition. This interaction process makes this flow similar to a swirling jet or a Batchelor vortex ($q$-vortex [16]). The loss of vortex structure can be explained by the development of an instability as in a $q$-vortex when the swirl parameter is low [17]. 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 $q \sim 1.01$ in case of the approach condition and $q \sim 0.4$ in case of the take-off condition. These values are below to the stability limit of $q$-vortices $0 < q_{lim} < 1.5$ [17]. The instability process takes place earlier in case of take-condition, but in both cases it leads to a turbulent flow.

The difference between the two configurations is well represented by the evolutions of the maximum azimuthal vorticity in Fig. 13, evaluated around the vortex core $\omega_{\Theta_{core}}$ and in the transverse plane $\omega_{\Theta_{peak}}$. One can observed that the interaction is more significant in case of take-off condition. Moreover, the evolution of cross-flow kinetic energy (Eq. 32) plotted in Fig. 14 shows at $\tau = 0.04$ a reduction of 20% in approach condition and 30% in take-off condition.

The circulation and velocity profiles of the vortex are reported in Fig. 15. The jet affects the vortex by reducing its peak of velocity and increasing its core. In case of approach condition, the vortex is not completely annihilated by the jet as show the tangential velocity profile at $\tau = 0.05$. Note that at
this time, the maximum tangential velocity has decreased of $\sim 55\%$ of its initial value. In both cases, the total circulation remains constant far from the vortex core.

Note the difference with a $q-$ or Batchelor vortex where all axial flow is concentrated in the core, axial momentum rapidly decays, leaving the vortex core almost unaffected, as show by the evolution of axial velocity isocontours in Fig. 16. The initial vortex position is also represented, but during the interaction it is slightly displaced.
3 Presentation of the numerical results

Fig. 12. Left side: approach condition and right side: take-off condition. Isosurfaces and isocontours of vorticity magnitude $\omega = \omega_{max}/e^3$. 
3 Presentation of the numerical results

**Fig. 13.** Evolutions of the maximum azimuthal vorticity around the vortex core $\omega_{\theta_{\text{peak}}}^{\text{core}}$ in dashed red line, and in the transverse plane $\omega_{\theta_{\text{peak}}}^{\text{transverse}}$ in black solid line. Left side and right side respectively for the approach and take-off condition.

**Fig. 14.** Evolution of the cross-flow kinetic energy $E_k$ during the jet/vortex interaction phase, in solid black line for the approach condition and in dashed blue line for the take-off condition.
Fig. 15. Circulation $\Gamma(r)$ and tangential velocity $v_\theta(r)$ profiles as a function of radial distance from the vortex center. Left side: approach configuration and right side: take-off condition.
Fig. 16. Left side: approach condition and right side: take-off condition. Isocontours of axial velocity averaged in axial direction (normalized by the aircraft velocity \( w_j = w/U \)). The initial vortex position and core radius are represented by the black solid line.
4 Conclusion

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

Two types of interaction 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 passive scalars 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 loss of vortex coherence. In case of approach condition the vortex is not completely annihilated contrary to the take-off condition. In both two cases, the jet affects the vortex by reducing its peak of velocity and increasing its core.

These results revealed that the vortex is very affected by the jet when it is close, and when the velocities ratio between the jet and vortex is high.
References

2. L. Nybelen, R. Paoli and G. Chevalier, Previous work and present knowledge on vortex interactions with jets and wakes, FAR-Wake, Deliverable D2.0, 2005