LES Modelling for Aerothermal Predictions Behind a Jet In Cross-Flow

J-C. Jouhaud*, L. Y. M. Gicquel * and B. Enaux †

CERFACS, 31057 Toulouse, France

M-J. Esteve‡

AIRBUS France, 31060 Toulouse, France

Abstract

Numerical investigations of a hot square Jet In Cross-Flow (JICF) issuing perpendicular to a flat plate are presented. This current work is initiated by AIRBUS France to better predict the aircraft aerothermal environment. Advanced numerical methods are tested in this specific context. In particular, the numerical approaches Large Eddy Simulations (LES) and Reynolds Average Numerical Simulations (RANS) are assessed in an industry-like configuration. Specific attention is brought to the LES of a JICF momentum flux ratio of 0.77 and a Reynolds number of 93,900 (based on cross flow velocity and jet diameter). Numerical issues such as grid resolution and boundary condition choices are addressed. Comparison of the Reynolds average quantities obtained from LES with RANS data at several cross-stream locations underlines the potential of LES for aerothermal predictions. Large scale motions observed experimentally and numerically are reproduced by the unsteady LES fields, and spectral analyses reveal the ability of the LES to reproduce the coherent structure frequencies.

Introduction

This work stems from the project MAEVA (Aerothermal Flow Modelling in Aircraft Ventilation) directed by AIRBUS France and ONERA with the co-operation of CERFACS. The aim of this project is to enhance the understanding and the prediction of the aircraft’s thermal environment in order to deal with the problems of composite materials. Indeed these materials, for an identical mass, are more resistant than the materials they replace. However
they are also more sensitive to the temperature variations and may age prematurely under extreme conditions. The MAEVA project is organized around three classes of problems:

- The impact of a hot jet on a cold wall,
- The mixed convection problem appearing in cavities,
- The interaction of a hot Jet In Cross-Flow (JICF) with a cold wall.

The purpose of this study is to assess Large Eddy Simulations (LES) for the third class of problems. Contrary to the Reynolds Average Navier-Stokes Simulations (RANS) technique which is restricted to steady turbulent flows, LES takes into account flow instabilities by solving for large flow structures while modelling the small scale effects. The implications on the numerical predictions are of importance but still need to be illustrated and validated in the context of industrial applications where thermal effects are of interest.

The investigation presented in this document aims at the modelling of the interaction between a hot JICF (see Margason\(^1\) for a detailed review on JICF’s and their applications) and the wall of a turbo-fan engine nacelle. The interaction takes place on the leading edge of the nacelle where hot air is injected through the wall of the nacelle in order to avoid icing on the exterior (see Fig. 1). The hot air is collected in a scoop and ejected on the leading edge of the nacelle to form the JICF. Since the velocities are of the order of tens of \(m.s^{-1}\), temperature can be considered as a passive scalar.

The objective of engine designers is to determine the essential characteristics of the jet and its development, and not to simulate the flow around the turbo-fan engine. The jet is thus studied in the simplified geometry of the flat plate configuration. This choice is justified by the fact that the nacelle has relatively low curvature and the jet diameter is very small in comparison with the dimension of the turbo-fan engine. The zone where the jet mixes with the cross flow can then be regarded as quasi-planar. Experimental data to be used in the validation of the numerical methods for this industry-like configuration were obtained using an apparatus located at ONERA. Velocity fields within the flow domain as well as thermal fields on the plate can be used to assess LES. The specific effect of the grid resolution on the LES predictions is also tested by performing computations on two meshes: a coarse grid and a fine grid.

This paper starts with a detailed presentation of the flow topology observed in free JICF. This is followed by a description of the LES methodology and a presentation of the computational domain. LES predictions are then compared with RANS predictions\(^2\) and experimental measures. The results are then discussed with a detailed description of the
mechanisms by which LES provides superior predictions to RANS in this aerothermal problem. To further analyse these encouraging results, coherent structures are investigated and spectral analyses are performed on the velocity signal obtained by LES.

Flow physics of a JICF

Flow features

The JICF configurations or transverse jets have been studied for many years because of their wide interest in fluid mechanics applications like V/STOL aircraft in transition flight, turbine film cooling, chimneys, roll control for rockets or fuel-mixing in combustion chamber.\textsuperscript{3,4} The JICF is a flow configuration that allows rapid mixing of two fluids thanks to complex three dimensional interactions. The results of these interactions between the jet and the transverse flow are coherent structures which were identified in the seventies.\textsuperscript{5,6} Nowadays, numerous investigations of these structures have been undertaken and they agree on four principal vortices (see Fig. 2):

1. The Counter-Rotating Vortex Pair (CVP):
   This structure is the dominant structure of the flow configuration far downstream of the jet orifice. It is formed by the deflection of the jet and is convected by the transverse flow. Experimental studies show that the CVP evolves from the reorientation of the shear layer vortices of the jet.\textsuperscript{7-10} This mechanism is confirmed by the numerical investigation of Cortelezzi and Karagozian.\textsuperscript{11} It is also important to notice that this is the main flow structure responsible for most of the mixing between the two fluids.

2. The horseshoe vortex:
   This is due to the adverse pressure gradient generated at the wall just ahead of the jet. This structure is quite similar to the structure observed for flows around a solid cylinder, and Krothapalli et al\textsuperscript{12} observed that the vortex shedding frequency is quite similar to that around a solid cylinder ($St = \frac{fD}{U_0} = 0.21$, where $D$ is the cylinder diameter, $U_0$ is the stream velocity and $f$ is the shedding frequency).

3. Shear layer ring vortices:
   These evolve on the jet circumference and are generated in the boundary layer of the jet orifice. These vortices become distorted when the jet is deflected and the mechanism of the shear layer evolution is interpreted by Kelso et al.\textsuperscript{13} The shear generated where the two streams meet (cross-flow and jet streams) creates a Kelvin-Helmholtz instability causing the flow to roll-up around the jet’s circumference. This roll-up creates vortex
tubes which extend up and around the jet, and which eventually shed into the main flow. These structures are named "hanging vortices" by Yuan et al.\textsuperscript{14}

4. The wake vortices: These structures have been studied experimentally\textsuperscript{7,15,16} but are the least understood. However, the experiments of Fric and Roshko\textsuperscript{16} suggest that this vorticity originates from the wall boundary layer where the boundary layer wraps around the jet. These flow patterns seem to be influenced by the jet trajectory.

**Correlations parameters**

A JICF can be characterized by mean quantity ratios involving velocity and density. The first parameter is the velocity ratio:

\[
V_R = \frac{U_{\text{jet}}}{U_{\text{cross}}},
\]

with:

- \(U_{\text{jet}}\): the jet bulk velocity
- \(U_{\text{cross}}\): the cross-stream velocity

If the density of the jet is different from the density of the cross flow, the velocity ratio Eq.( 1) no longer describes the jet centerline evolution. The following parameter,\textsuperscript{17} is thus preferred:

\[
C_R = \frac{\rho_j U_{\text{jet}}}{\rho_0 U_{\text{cross}}}
\]

A last parameter shows better parameterization properties\textsuperscript{18} in the case of significant thermal variations and is the square-root of the momentum flux ratio:

\[
R = \sqrt{\frac{\rho_j U_{\text{jet}}^2}{\rho_0 U_{\text{cross}}^2}}
\]

In the present case \(C_R = 0.7\) and \(R = 0.9\). For these low values, the jet is expected to remain within the cross flow boundary layer. Such jet velocity ratio (\(C_R < 1\)) have been studied experimentally\textsuperscript{19} and numerically\textsuperscript{20} on round JICF. However, no published data on such jet velocity ratios deals with square JICF. Renze et al.\textsuperscript{20} studied the impact of the density ratio on angled jets, and Andreopoulos\textsuperscript{19} investigated a jet issuing perpendicularly but with a density ratio equal to unity.
Numerical Simulation: LES

The aim of the LES is to resolve the large scale of turbulence while the smaller ones are modeled based on their universality. In contrast, RANS attempts to model all the flow scales at once. Consequently, LES are better suited to industrial configurations where large scales are known to be essential. In the case of the JICF, where the unsteady behaviour of the various flow structures is expected to be important, the unsteady LES approach which provides spatio-temporal resolution should be well adapted.

Governing equations

A filtering operation is applied to the continuity equation, the transport equations of momentum, total energy and species. In the case of compressible flows a Favre filtering\textsuperscript{21} is used:

\[
\overline{\rho f}(x, t) = \rho f(x, t) = \int_{-\infty}^{+\infty} \rho f(x', t) \, \mathcal{G}(x' - x) \, dx',
\]

where \(\mathcal{G}\) is the filter function, \(\rho\) is the density and \(f\) can be either the velocity vector, the total energy or the mass fraction of species. The application of the filtering operation to the set of governing compressible Navier-Stokes equations yields:\textsuperscript{22-24}

\[
\begin{align*}
\frac{\partial \overline{\rho}}{\partial t} + \frac{\partial}{\partial x_j}(\overline{\rho} \, \overline{u}_j) &= 0, \\
\frac{\partial \overline{\rho} \, \overline{u}_i}{\partial t} + \frac{\partial}{\partial x_j}(\overline{\rho} \, \overline{u}_i \, \overline{u}_j) &= -\frac{\partial}{\partial x_j} [\overline{P} \, \delta_{ij} - \overline{\tau}_{ij} - \overline{\tau}^i_j], \\
\frac{\partial \overline{E}}{\partial t} + \frac{\partial}{\partial x_j}(\overline{\rho} \, \overline{E} \, \overline{u}_j) &= -\frac{\partial}{\partial x_j} [u_i (P \, \delta_{ij} - \overline{\tau}_{ij}) + \overline{q}_j + \overline{q}^j].
\end{align*}
\]

where,

\[
\overline{\tau}_{ij} = 2\mu (S_{ij} - \frac{1}{3} \delta_{ij} S_{kk}),
\]

\[
\approx 2\overline{\mu} (\tilde{S}_{ij} - \frac{1}{3} \delta_{ij} \tilde{S}_{kk}),
\]

\[
\overline{q}_i = -\lambda \frac{\partial \overline{T}}{\partial x_i},
\]

\[
\approx \lambda \frac{\partial \overline{T}}{\partial x_i}
\]

and,

\[
\tilde{S}_{ij} = \frac{1}{2} (\partial \overline{u}_j / \partial x_i + \partial \overline{u}_i / \partial x_j).
\]

This set of partial differential equations is suplemented by the ideal gas law \(\overline{P} = \overline{\rho} \, r \, \overline{T}\) where \(r\) is the gas constant.
Sub-grid scale model

In Eq. (5), the terms $\tau_{ij}$ and $q_{ij}$ need to be modeled to close the set of LES equations. To do so, the Boussinesq hypothesis Eq. (8) is introduced to mimic the sub-grid scale action on the resolved field:

$$\tau_{ij} = -\rho \left( \overline{u_i u_j} - \overline{\tilde{u}_i \tilde{u}_j} \right) = 2 \nu_t \left( \tilde{S}_{ij} - \frac{1}{3} \tilde{S}_{kk} \delta_{ij} \right) + \frac{1}{3} \tilde{r}_{kk} \delta_{ij},$$

where the sub-grid scale turbulent viscosity $\nu_t$ needs to be given. Here the standard Smagorinsky\textsuperscript{25} model is used:

$$\nu_t = (C_S \Delta)^2 \sqrt{2 \tilde{S}_{ij} \tilde{S}_{ij}},$$

where $\Delta$ denotes the filter characteristic length (cube-root of the cell volume), $C_S$ is the standard model constant set to 0.18. Finally, the thermal turbulent heat flux, $q_{ij}$, is closed through the use of the turbulent Schmidt number, $Pr_t = 0.9$ along with a gradient hypothesis.

Flow solver

In the present work, the fully compressible LES equations are solved on unstructured grids composed of tetrahedral cells in a code called AVBP\textsuperscript{26}. For this code, the sub-grid scale turbulence is modelled with the Smagorinsky model and with wall-functions\textsuperscript{27}. The numerical implementation is based on the classical centered finite-volume cell-vertex space discretization of Lax-Wendroff. For time integration, the explicit 4-stages Runge-Kutta scheme is used with CFL=0.7.

Flow domain

As illustrated in Fig. 3, hot air is injected into tubes (Inlets 1 and 2) which reach a settling chamber called a scoop. The scoop is covered on the top by a fine copper plate denoted by grid. The hot air mixes in the scoop and is ejected vertically into the cold cross-stream which flows over the flat plate. The width of the injection orifice, $D$, measures 30 mm and the jet exit is flush with the horizontal flat plate.

The jet orifice is located $X/D = 2.8$ downstream the domain inlet, and $X/D = 33.3$ upstream of the outlet. Each lateral face of the computational domain is located $Y/D = 27.8$ from the jet orifice, and the domain size used for the computation is $37.2D \times 46.5D \times 33.3D$. The spreading of the jet is not restricted by the domain.

Two grids (Fig. 5) are used for the simulation: a coarse mesh $M_1$ containing 580,000 cells and a finer mesh $M_2$ composed of 2,740,000 cells. Each unstructured mesh is composed of tetrahedral cells and grid refinement is enforced along the jet trajectory and in the near wall.
region. $M_1$ and $M_2$ have the same topology and they only differ in their mesh density (the finest mesh is locally two times more refined), in the region of the JICF interaction and inside the scoop (see insets on Fig. 5). There are respectively around 150 points ($\Delta x \approx \Delta y \approx 0.2$ mm) and 300 points ($\Delta x \approx \Delta y \approx 0.1$ mm) in the square jet orifice for the coarse mesh and the fine mesh. These two computational grids follow the approach used for similar flows$^{28,29}$ and they permit a good representation of the interaction.

The flow to be simulated corresponds to the MAEVA$^2$ experimental test-case (Mod- elization of Aerothermal Flows for Aircraft Ventilation). It is characterized by a cross-flow reynolds number of 93,900 (based on the injection hydraulic diameter $D$). The boundary layer thickness on the flate plate seen by the cross-flow is $\delta/D = 0.13$. To allow a manageable number of cells, a law-of-the-wall model is used at the bottom surface. In both grids the cells adjacent to the bottom wall have sizes corresponding to $\Delta x^+, \Delta y^+, \Delta z^+ \gg 1$ ($\Delta x^+ = 110, \Delta y^+ = 150, \Delta z^+ = 130$ for $M_2$, $\Delta x^+ = 420, \Delta y^+ = 450, \Delta z^+ = 460$ for $M_1$). Use of a law-of-the-wall is justified here because $\delta/D = 0.13$: the inlet turbulence is one order of magnitude under the turbulence generated by the jet.

**Boundary conditions**

In accordance with the experimental measurements, the flow through the main inlet is imposed by using a velocity profile which includes a boundary layer at the wall. The profile was chosen to match exactly the experimental profile (see Fig. 4) including the log-layer region. In this way, we expect to simulate the correct boundary layer thickness before the jet interaction.

The initial boundary layer height is $2D$ and the free-stream velocity is $U_0 = 47.1\ m.s^{-1}$. The injection pipes which feed into the scoop are given mean turbulent pipe profiles according to the standard $1/7$ power law approximation (Table 1). All these velocity profiles agree with the mass flux of the experiments as well as the temperature conditions. Because of the acoustic waves present in LES and inherent to the compressible flow solver, characteristic boundary conditions$^{30,31}$ are applied at all inlets, with relaxation coefficients$^{32}$ on the velocity, the temperature and the mass fraction of the species. The same type of characteristic boundary condition is applied at the outlet but only pressure is specified. The bottom plate is divided in two parts. The "grid" region immediately surrounding the jet orifice uses either a no-slip isothermal wall law condition ($M_2$) or a no-slip adiabatic wall law condition ($M_1$ and $M_2$). The remaining area of the plate is treated by use of a no-slip adiabatic wall law condition. At the lateral boundaries, symmetry conditions are enforced, and a slip adiabatic wall condition is imposed on the top surface of the computational domain. This last condition is valid in our case because it is located sufficiently far from the jet exhaust ($33.3D$ above
the plate). All the boundary conditions used for LES are summarised in Table 2.

**Results**

*Comparison with experiment and RANS predictions*

In this section, a comparison is made between the flow field results of the experiment, RANS, and LES. The experimental measurements\(^2\) are obtained using PIV (Particle Image Velocimetry, Exp 50036) and LDA (Laser Doppler Anemometry, Exp 50037) on several cross-stream sections, perpendicular to the jet. In addition to this database, hot wire measurements (Exp 11300) are provided for the heated jet in the form of temperature maps.

The RANS calculation was performed with the elsA\(^33\) code of AIRBUS. The characteristics of the simulation are summarised below:

- a structured mesh composed of 5,900,000 cells with the first cell adjacent to the bottom wall corresponding to \(\Delta y^+ \approx 1\),
- the \(k-l\) two equation turbulence model of Smith,\(^{15}\)
- a cell-center finite-volume method,
- an implicit method for time integration (CFL=3000.).

All the results obtained by LES are time-averaged to be directly comparable to the RANS and experimental data. Time integration is in this case carried out over a duration equal to 2.5 flow-through times with a flow-through time being obtained based on the cross-stream bulk velocity and the computational length (i.e. 47.1 m.s\(^{-1}\) and 37.2\(D\)). This duration is sufficient for convergence of all average quantities.

The predictions are assessed on velocity and kinetic energy profiles at three cross-planes respectively located at \(X/D = 1, 3\) and 8 downstream of the jet exit. Furthermore, comparison of the cooling effectiveness \(\eta\) allows an assessment of the potential for each approach to predict the thermal process as well as the mixing behaviour. In the following the cooling effectiveness is defined by:

\[
\eta = \frac{<T_w> - T_0}{T_j - T_0}
\]

with:

- \(<T_w>\): the temporally averaged local wall temperature
- \(T_j\): the jet temperature
- \(T_0\): the temperature of the cross flow upstream of the jet exit
Profiles of $\eta$ are presented for different streamwise locations ($X/D = 1, 8, 14$) and one more profile is plotted in the spanwise symmetry plane ($Y/D = 0$).

The effect of the boundary condition to be applied at the "grid" is also addressed in terms of $\eta$ profiles. Indeed the exact thermal boundary condition to be used in this wall region is not clear because of the composition of the "grid": $1\text{mm}$ thick copper plate. To investigated its importance, two simulations are carried out using mesh $M_2$ only: one LES where the "grid" is considered as adiabatic and a second LES where the "grid" is considered as isothermal with a "grid" temperature approximated to the average temperature between the two grid sides. The following section illustrates the importance of this boundary condition for the proper prediction of the cooling effectiveness.

1. The coarse grid results

In this simulation the "grid" is considered as adiabatic. An instantaneous visualization of the temperature field (Fig. 6) shows that the cells are too coarse to properly represent the coherent structures located in the shear layer. However, velocity profiles presented in Fig. 7, show a rather good correspondance between LES/RANS/Experiments except maybe at the cross-plane $X/D = 1$. In fact, the measured reverse-flow is not well-predicted by either simulation. This back-flow is not captured because of the wall grid resolution and modelling of LES which seems inadequate.

With regard to the prediction of mixing, the cooling effectiveness profiles predicted by LES in Fig. 8 are under-estimated wether they are in the streamwise locations or in the spanwise symmetry plane. The under-estimation of the mixing behaviour is also evidenced by the velocity field comparison (Fig. 9). The vertical velocity vector component $V/U_0$ shows that the CVP predicted by LES is not as strong as it should be. The field $U/U_0$ shows also that the jet penetration is not well reproduced. LES tend to over-estimate the jet penetration which can in part be explained by incorrect jet/wall interaction caused by the poor grid resolution in the near wall region.

2. The fine grid results

To assess the effect of mesh refinement along the jet trajectory, a new simulation is conducted based on mesh $M_2$. The first LES used a no-slip adiabatic wall law condition while a second LES used a no-slip isothermal wall law on the "grid".

The comparison between LES predictions and measured data shows very good agreement and clearly illustrate the impact of the grid resolution on the LES predictions, Fig. 6. Thanks to refinement, definite structures are observed which are not dissipated.
excessively resulting in enhanced prediction of the mixing process. The velocity field $V/U_0$ illustrated in Fig. 10 shows that the intensity of the CVP is well predicted. Similarly, the jet penetration is improved when compared to the mesh $M_1$ predictions and the $M_2$ results better reproduce the experimental observations.

The fine grid LES captures more coherent structures of the JICF which should directly impact the predictions of cooling effectiveness (Fig. 11). The $\eta$ profile in the spanwise symmetry plane of Fig. 11 is well predicted and RANS or LES equally well reproduce the experimental measurements. Given the length of the thermal trace, this type of profile is however relatively easy to obtain. On the other hand, transverse profiles, given the width of the thermal trace, are more difficult to predict. In that specific respect, LES gives good agreement with the experimental database contrary to RANS which diffuses too much. Fig. 11 also shows that the isothermal wall boundary condition applied to the "grid" enhances the $\eta$ prediction in the $X/D = 1$ plane. The thermal transfer with this boundary condition are better predicted than with the adiabatic "grid". To improve even further the $\eta$ prediction, the next step would be to solve the heat transfer within the "grid" plate.

The velocity profiles predicted by LES (Fig. 12) do not display major differences between the coarse and fine simulations. As already mentioned for the coarse grid, the back-flow region is still not well-predicted as illustrated by the profile in the cross-plane $X/D = 14$ (Fig. 12). The origin of this deficiency appears to be the wall law treatment, which may be inadequate in this specific region. With regard to the other planes, the velocity is slightly better resolved very close to the wall.

The resolved turbulent kinetic energy (TKE) of the LES, $k = \frac{1}{2} < u'_i u'_i >$ where $u'_i = (\tilde{u}_i - < \tilde{u}_i >)$ is also presented on Fig. 12 and compared to the measured data. The profiles show that the TKE is under-estimated in the region next to the wall. This is explained by the fact that only the resolved scales contributing to the turbulent kinetic energy are available here and the sub-grid contribution of the LES model does not appear. In the wall region, the shear generated small scales of turbulence may not be represented properly by the mesh and the wall law model, this can also explain the under-estimation. On the contrary, far away from the wall, a region dominated by very large scales, the TKE is resolved and good agreement between LES and measurements is observed. Note however that the standard Smagorinsky model overestimates the TKE far from the wall, which is a result already observed by Schlüter and Schönfeld.34
Instantaneous flow filed analysis

To illustrate the ability of LES to reproduce the interaction between the two fluids, and resulting from the coherent structures, the detection criteria $Q$ (Hunt et al.\textsuperscript{35}) is introduced in the flow visualization. This specific diagnostic is justified by the fact that for our specific problem where the jet remains close to the wall, it is better than the flow vorticity. Indeed, high vorticity marks the locations of coherent structures but also the locations of high shear. Next to the wall the shear is important and prevent proper identification. To cure this default, the criteria $Q$ is tested. The coherent structures are plotted through an iso-surface in Fig. 13. An arch-like coherent structure is clearly evidenced just above the jet exhaust. This pattern is observed by Andreopoulos\textsuperscript{19} in his experimental flow visualizations and by Yuan and Street\textsuperscript{14} in their LES.

Thanks to animations, one observes that this structure is shed into the main flow and positions itself above the CVP. The apparition of such a flow instability seems in agreement with Kelvin-Helmholtz instability. Moreover, and based on the temperature as a passive scalar, it appears that the flow transported by the Kevin-Helmholtz structures is in the most part issued from the scoop.

The visualization also reveals that the detachment of these structures occurs on the lateral edges of the jet exit. This phenomenon is also reported by Yuan and Street\textsuperscript{14} who propose two mechanisms to explain the "breakdown process":

- an adverse pressure gradient on the lateral edges,
- a strong upstream flow in the wake of the jet.

After the "breakdown process", these structures composed of hot air, are reoriented, stretched and finally, within a short distance of roughly 6-10 D, diffused by the action of the turbulent shear stresses and viscosity. Sometimes these structures impact the planar plate leading to a crucial aerothermal process: heat is thus brought to the plate. It explains the prediction difference on the wall cooling effectiveness obtained by RANS and LES.

Spectral Analysis

The flow field animation and the effects of mesh refinement observed above reveal that capturing the coherent structures with LES is a paramount to prediction of the proper mean and fluctuating quantities. It also explains the poor RANS predictions of the temperature profiles since it does not simulate the unsteadiness responsible for the mixing.

To determine the role of the structures observed in Fig. 13, spectral analyses of various quantities are presented below. First, the vortex shedding behind the jet column is analysed and observed in the LES animations. To validate the LES approach, results are compared
to the experimentally measured frequencies obtained by Fast Fourier Transform (FFT) of
the velocity signal recorded by hot wire probes. Subsequently, and after the validation of
the frequency acquisition method, LES predictions are assessed against the experimental
frequency measured in the shear region.

In this paragraph, the frequency acquisition employed to study LES results is explained.
Probes are located in the shear and wake regions in order to characterize the vortices in these
areas (Fig. 14). FFT’s are performed on the velocity signals using Welch’s method.\(^\text{36}\) It con-
sists of dividing the time series data in segments, computing a periodogram of each segment
and averaging the power spectral densities. In addition to this first step, the segments are
overlapped to artificially increase the overall duration of each signal, and windows (Hanning,
Blackman) are used to remove erroneous frequencies from the periodogram introduced by
these manipulations (i.e. frequentional resolution, dynamic resolution). Averaging is used to
reduce the variance of the estimate and overlap is introduced to decrease bias.

The parameters used for Welch’s method summarised in Table 3. To give a better view
of the performance of this acquisition method, a comparison is made with a conventional
FFT approach in Fig. 15. The results clearly illustrate the decrease of variance thanks to
the Welch Method and a problem which occurs when the time series are too short.

In the set of results presented below the duration of each segment equals 0.032 s and
the probes see about 7 convective times which is sufficient to trust the statistics on the
periodogram. For the wake region the spectral analysis is based on the time series of the
axial velocity component \(\tilde{u}(t)\) obtained on the fine grid. For the shear region, the time series
of the vertical velocity component \(\tilde{w}(t)\) are used.

Fig. 16 shows the LES predictions for the probes located in the wake region downstream
the jet nozzle. LES predicts vortex shedding at a frequency corresponding to a range of
Strouhal numbers \(0.13 < St < 0.16\) whereas the experiment brings out a Strouhal number
equal to 0.15 \((St = \frac{fD}{U_0}\), where \(D\) is the orifice size, \(U_0\) is the cross flow velocity and \(f\) is
the shedding frequency). It is difficult to draw conclusions from comparisons with Strouhal
number found in the literature since the amount data is limited, especially for square jets, and
because the kinetic energy ratio in this study is very low \(R = 0.77\). However it is interesting
to note Moussa and Trischka\(^7\) also detected a wake Strouhal number approaching 0.15 in
flows with \(V_R\) ranging from 2 to 8 and with the Reynolds number fixed to 8,000. Therefore
this study underlines the independence of the wake shedding on the Reynolds number which
confirms the work of McMahon et al.\(^{37}\)

This analysis of the vortex shedding frequency further proves the ability of LES to re-
produce the flow physics. In order to complete the experimental database and investigate
the coherent structures illustrated on Fig. 14, the shear layer region is investigated. In con-
trast with the wake, this region of the flow and particularly its modes are more precisely documented by researchers. Improving the jet penetration and its spread is one of the justifications for this interest. In the present study, two probes are positioned above the upstream edge of the jet exit to characterize the structure frequency. Figure 17 shows velocity spectra in the shear layer region of the JICF. The spectra depict a peak corresponding to a Strouhal number equal to 0.2 and higher harmonics of the vortex roll-up frequency are observed. The spectral peak is also more distinct when the measurement is not performed close to the jet exit. This observation is confirmed by the work of Narayanan et al. However, they also notice a decrease of the frequency value when the probe moves inside the jet, a behaviour that is not observed in the present study. The results on the shear layer region show that the coherent structure is relatively energetic. These results based on the shear layer region confirm the importance of these coherent structures which are important if proper mixing is to be predicted by LES.

Conclusions

Large Eddy Simulations (LES) are conducted to study a jet flow which is found on the leading edge of a turbo-fan engine nacelle, part of a work initiated by AIRBUS France. The aim of the study is to demonstrate the ability of this numerical method to reproduce aerothermal predictions in an industrial context. Simulations are thus conducted of a hot Jet In Cross-Flow issuing from a square nozzle, and the results are compared with those of Reynolds Average Numerical Navier-Stokes Simulations (RANS) and experimental measurements. The quality of the LES predictions of cooling effectiveness are of particular interest.

The present Large Eddy Simulations of a jet in crossflow reproduce with good agreement the experimental measurements, in contrast with Reynolds Average Navier-Stokes Simulations. Large Eddy Simulation identifies several unsteady structures in the near field of the jet which are not predicted by Reynolds Average Navier-Stokes Simulations. These structures are essential for the mixing predictions downstream of the jet, that is why particular attention must be given to the Large Eddy Simulation mesh resolution.

This work also reveals that the ”grid” thermal boundary condition is significant, and improvements in the Large Eddy Simulation predictions might require solution of the heat transfer within the grid plate. Such a task may be considered for future developments since coupling a computational fluid dynamics solver with a thermal code is conceivable thanks to computational frameworks such as PALM for example.

Finally, spectral analyses of the wake and shear layer regions demonstrate the quality of the Large Eddy Simulation’s predictions obtained here and further illuminate the unsteady flow features. However and as of today, current Large Eddy Simulation’s applications remain
limited from an aerothermal point of view because of the very high Reynolds number usually involved (about $Re = 60,000,000$). Nonetheless, this work proves the potential of Large Eddy Simulation for this type of problem with important industrial implications.

**Acknowledgments**

The authors gratefully acknowledge AIRBUS France for its support.

**References**


List of Tables

1  Set of reference values applied at the boundary conditions of LES.  18
2  Boundary condition specification. ................................. 19
3  Welch’s method characteristics. ................................ 20
Table 1: Set of reference values applied at the boundary conditions of LES.

<table>
<thead>
<tr>
<th>Boundary name</th>
<th>Velocity ((m.s^{-1}))</th>
<th>Temperature ((K))</th>
<th>Pressure ((Pa))</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet1</td>
<td>(</td>
<td>V_1</td>
<td>= 53.1)</td>
</tr>
<tr>
<td>Inlet2</td>
<td>(</td>
<td>V_2</td>
<td>= 53.1)</td>
</tr>
<tr>
<td>Inflow</td>
<td>(U_0 = 47.1)</td>
<td>(T_0 = 293)</td>
<td>(P_0 = 98,701)</td>
</tr>
</tbody>
</table>
Table 2: Boundary condition specification.

<table>
<thead>
<tr>
<th>Boundary name</th>
<th>Fixed quantities</th>
<th>Relaxation coefficient</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet1</td>
<td>( U, V, W, T, Y )</td>
<td>( \sigma_{U,V,W,T,Y} = 50 )</td>
</tr>
<tr>
<td>Inlet2</td>
<td>( U, V, W, T, Y )</td>
<td>( \sigma_{U,V,W,T,Y} = 50 )</td>
</tr>
<tr>
<td>Inflow</td>
<td>( U, V, W, T, Y )</td>
<td>( \sigma_{U,V,W,T,Y} = 50 )</td>
</tr>
<tr>
<td>Outflow</td>
<td>( P )</td>
<td>( \sigma_p = 50 )</td>
</tr>
<tr>
<td></td>
<td>( T )</td>
<td></td>
</tr>
<tr>
<td>Scoop</td>
<td>( q_{\text{wall}} = 0 )</td>
<td>NA</td>
</tr>
<tr>
<td>Bottom wall</td>
<td>( q_{\text{wall}} = 0 )</td>
<td>NA</td>
</tr>
<tr>
<td>Lateral surfaces</td>
<td>SYMMETRY</td>
<td>NA</td>
</tr>
<tr>
<td>Top surface</td>
<td>( q_{\text{wall}} = 0 )</td>
<td>NA</td>
</tr>
</tbody>
</table>
Table 3: Welch’s method characteristics.

<table>
<thead>
<tr>
<th>Time series duration</th>
<th>Number of segments</th>
<th>Window</th>
<th>Overlap</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.144 s</td>
<td>8</td>
<td>Hanning or Blackman</td>
<td>50%</td>
</tr>
</tbody>
</table>
List of Figures

1. Schematic illustration of the engine anti-icing system .......................... 22
2. Schematic illustration of the vortex system in a JICF\textsuperscript{16} .................. 23
3. Computational domain considered for LES: (a) Top view and, (b) Lateral view, and (c) Front view. .................................................. 24
4. Velocity profile imposed in the main inlet ($X/D = -4.0$, coarsest mesh). .......................................................... 25
5. Cross view of the mesh refinement in the $Y/D = 0$ plane: a) the coarse mesh $M_1$, and b) the fine mesh $M_2$. .......................... 26
6. Instantaneous view of temperature (white=hot, black=cold) as obtained by LES in the $Y/D = 0$ plane: a) Coarse mesh $M_1$, and b) Fine mesh $M_2$. .......................................................... 27
7. Comparison of the velocity profiles in the symmetry plane at different streamwise locations (simulation on mesh $M_1$). .......................... 28
8. Comparison of the wall cooling effectiveness profiles at different streamwise locations and in the spanwise symmetry plane ($Y = 0$) (simulation on mesh $M_1$). .......................................................... 29
9. Comparison of the velocity fields at $X/D = 1$ (simulation on the mesh $M_1$). .......................................................... 30
10. Comparison of the velocity fields at $X/D = 1$ (simulation on the mesh $M_2$). .......................................................... 31
11. Comparison of the wall cooling effectiveness profiles at different streamwise locations and in the spanwise symmetry plane ($Y = 0$) (simulation on mesh $M_2$). .......................................................... 32
12. Comparison of the velocity profiles and turbulent kinetic energy profiles in the symmetry plane at different streamwise locations (simulation on mesh $M_2$). .......................................................... 33
13. Coherent structures indicated by a $Q$ iso-surface ($Q = 2.1 \times 10^7$). ........... 34
14. Probe locations: a) Wake region (top view), b) Shear layer region (lateral view). .......................................................... 35
15. Comparison between a conventional FFT approach and the Welch method. .......................................................... 36
16. Axial velocity spectra measured at six locations downstream of the jet exit. .......................................................... 37
17. Vertical velocity spectra measured at two locations along the jet trajectory. .......................................................... 38
Fig. 1: Schematic illustration of the engine anti-icing system
Fig. 2: Schematic illustration of the vortex system in a JICF\textsuperscript{16}. 

Crossflow

Jet Shear Layer Vortices

Counter Rotating Vortex Pair

Horseshoe Vortex

Wake Vortices
Fig. 3: Computational domain considered for LES: (a) Top view and, (b) Lateral view, and (c) Front view.
Fig. 4: Velocity profile imposed in the main inlet ($X/D = -4.0$, coarsest mesh).
Fig. 5: Cross view of the mesh refinement in the $Y/D = 0$ plane: a) the coarse mesh $M_1$, and b) the fine mesh $M_2$. 
Fig. 6: Instantaneous view of temperature (white=hot, black=cold) as obtained by LES in the \( Y/D = 0 \) plane: a) Coarse mesh \( M_1 \), and b) Fine mesh \( M_2 \).
Fig. 7: Comparison of the velocity profiles in the symmetry plane at different streamwise locations (simulation on mesh $M_1$).
Fig. 8: Comparison of the wall cooling effectiveness profiles at different streamwise locations and in the spanwise symmetry plane ($Y = 0$) (simulation on mesh $M_1$).
Fig. 9: Comparison of the velocity fields at $X/D = 1$ (simulation on the mesh $M_1$)
Fig. 10: Comparison of the velocity fields at $X/D = 1$ (simulation on the mesh $M_2$)
Fig. 11: Comparison of the wall cooling effectiveness profiles at different streamwise locations and in the spanwise symmetry plane ($Y = 0$) (simulation on mesh $M_2$).
Fig. 12: Comparison of the velocity profiles and turbulent kinetic energy profiles in the symmetry plane at different streamwise locations (simulation on mesh $M_2$).
Fig. 13: Coherent structures indicated by a $Q$ iso-surface ($Q = 2.1 \times 10^7$).
Fig. 14: Probe locations: a) Wake region (top view), b) Shear layer region (lateral view).
Fig. 15: Comparison between a conventional FFT approach and the Welch method.
Fig. 16: Axial velocity spectra measured at six locations downstream of the jet exit.
Fig. 17: Vertical velocity spectra measured at two locations along the jet trajectory.