This item was submitted to [Loughborough's Research Repository](#) by the author.
Items in Figshare are protected by copyright, with all rights reserved, unless otherwise indicated.

Large-eddy simulation of twin impinging jets in cross-flow

PLEASE CITE THE PUBLISHED VERSION

<http://aerosociety.com/Assets/Docs/Protected/Subscribers/AeroJournal/3097.pdf>

PUBLISHER

© Royal Aeronautical Society

VERSION

VoR (Version of Record)

LICENCE

CC BY-NC-ND 4.0

REPOSITORY RECORD

Li, Q, Gary J. Page, and James J. McGuirk. 2012. "Large-eddy Simulation of Twin Impinging Jets in Cross-flow". figshare. <https://hdl.handle.net/2134/9499>.

This item was submitted to Loughborough's Institutional Repository (<https://dspace.lboro.ac.uk/>) by the author and is made available under the following Creative Commons Licence conditions.

For the full text of this licence, please go to:
<http://creativecommons.org/licenses/by-nc-nd/2.5/>

Large-eddy simulation of twin impinging jets in cross-flow

Q. Li, G. J. Page and J. J. McGuirk

Loughborough University

Department of Aeronautical and Automotive Engineering

Leicestershire, UK

ABSTRACT

The flow-field beneath a jet-borne vertical landing aircraft is highly complex and unsteady. Large-eddy simulation is a suitable tool to predict both the mean flow and unsteady fluctuations. This work aims to evaluate the suitability of LES by applying it to two multiple jet impingement problems: the first is a simple twin impinging jet in cross-flow, while the second includes a circular intake. The numerical method uses a compressible solver on a mixed element unstructured mesh. The smoothing terms in the spatial flux are kept small by the use of a monitor function sensitive to vorticity and divergence. The WALE subgrid scale model is utilised. The simpler jet impingement case shows good agreement with experiment for mean velocity and normal stresses. Analysis of time histories in the jet shear layer and near impingement gives a dominant frequency at a Strouhal number of 0.1, somewhat lower than normally observed in free jets. The jet impingement case with an intake also gives good agreement with experimental velocity measurements, although the expansion of the grid ahead of the jets does reduce the accuracy in this region. Turbulent eddies are observed entering the intake with significant swirl. This is in qualitative agreement with experimental visualisation. The results show that LES could be a suitable tool when applied to multiple jet impingement with realistic aircraft geometry.

NOMENCLATURE

Δ	filter width
Γ	cell volume
μ_t	it subgrid scale viscosity
$\nabla \cdot \mathbf{u}$	divergence of velocity
Ω	vorticity
ρ	density
$\varepsilon_1, \varepsilon_2, \varepsilon_3$	smoothing parameters
A_{ij}	flux Jacobian
C_s	Smagorinsky model constant
C_w	WALE model constant
D_j	jet diameter
E	total energy per unit volume
f	frequency
$\mathbf{F}, \mathbf{G}$	flux vectors
L	pseudo-Laplacian
$\mathbf{n}$	normal vector
p	pressure
$\mathbf{Q}$	pressure vector
Q_ε	positive invariant of velocity divergence
R	jet to cross-flow velocity ratio
S_{ij}	strain rate tensor
St	Strouhal number

t	time
T_{sc}	scalar temperature
u, v, w	Cartesian components of velocity
U_c	cross-flow velocity magnitude
U_n	face normal velocity component
W_j	jet velocity magnitude
x, y, z	Cartesian components of position

1.0 INTRODUCTION

There is an increasing demand from industry to apply large-eddy simulation (LES) to flow problems where information on the fluctuating unsteady behaviour is as important as the mean flow field. An example of such a flow problem is the in-ground-effect phase of jet-borne Short-Take-Off/Vertical Landing (STOVL) aircraft.

The flowfield surrounding the aircraft during this mode of operation is a complex interaction of lift jet ground impingement, upwash fountain flow and ground vortex flow (Fig. 1). If hot gas enters the engine intake compressor stall, engine surge and performance loss may occur^(1,2). This phenomenon is normally referred to as Hot gas ingestion (HGI). The multiple lift jets impinge upon the ground plane forming wall jets, these then collide and produce an upward flow, or 'upwash fountain'. This fountain can potentially direct hot gas towards the engine intakes and devices under the fuselage may be required to prevent hot gas ingestion. A second mechanism for HGI is the forward motion of the aircraft, coupled with the oncoming wind, which rolls up the forward flowing wall jet to produce a ground vortex. Again, this can lead to hot gas entering the intake, although the path is relatively long and temperatures are likely to be less than the more direct fountain route.

Solutions using the Reynolds-averaged Navier-Stokes (RANS) equations with a turbulence model can provide reasonable predictions of the large ground vortex and turbulent mean flow^(3,4,5), but are unable to provide information on important phenomena such as the unsteady fountain and instantaneous flow distortion and swirl in the intake. The ability to predict both the mean flow and unsteady excursions is important for the design and development of future aircraft.

When considering experimental and computational studies, an important parameter is the jet to cross-flow velocity ratio as this controls the location of the ground vortex. Similarly, the fountain strength is a strong function of the height of the impingement plane and the separation of the jets, both normalised by jet diameters.

Barata *et al*⁽⁶⁾ carried out experiments in a water tunnel with a jet to cross-flow ratio of 30, for a Reynolds number based on the jet exit diameter and velocity Re_j between 60,000 and 105,000, and with the jet exit five jet diameters above the ground plane. Measurements were conducted using Laser Doppler Anemometry (LDA). Behrouzi and McGuirk⁽⁷⁾ investigated the intake ingestion problem with LDA. Three typical jet to cross-flow ratios of 18, 24, 35 relevant to vertical landing aircraft hot gas ingestion were chosen. The Reynolds number in terms of jet centreline velocity and diameter was 40,000. In Barata *et al*⁽⁶⁾ and Behrouzi and McGuirk⁽⁷⁾, the mean velocity and turbulence quantities at different locations were measured to serve as a benchmark validation database for numerical simulations. Consequently, in the current work these two test cases will be used to validate the capability of large-eddy Simulation.

Flow field and noise characteristics of a supersonic impinging jet were studied by Krothapalli *et al*⁽⁸⁾. Suck down force or lift loss was investigated and was found to increase as the ground plate approached the nozzle exit due to the entrainment flow associated with the lifting jets. The presence of the ground plane increased the overall sound pressure level (OASPL) by approximately 8dB relative to a corresponding free jet. Although there was no cross-flow in Cabrita *et al*'s⁽⁹⁾ twin-jet experiment, its high speed visualisation still gives important information on the unsteady fountain and jet shear layer variations. Active and passive control of supersonic impinging single jet and twin jets have been studied by Alkislar *et al*⁽¹⁰⁾ and Lou *et al*⁽¹¹⁾ using microjets. Nearfield microphone spectra suggested that, for the non-controlled case, there were dominant frequencies in the jet shear-layer or close to the ground plate.

The potential usefulness of LES for the prediction of the upwash fountain was shown in the 1980s with pioneering work by Childs and Nixon⁽¹²⁾, Childs *et al*⁽¹³⁾ and Rizk and Menon⁽¹⁴⁾. Although limited by available computing power, these calculations showed how the collision of the turbulent wall jets could create a highly unsteady fountain that was in general agreement with experimental studies. More recently, computations by Chaderjian *et al*⁽⁴⁾ and Pandya *et al*⁽⁵⁾ showed how an unsteady RANS method could predict the large scale unsteadiness in the ground vortex, but this type of method will not improve the prediction of the near field (particularly the impingement and the fountain flow). Page *et al*⁽¹⁵⁾ used a structured finite volume, pressure-based LES technique to compute two simplified twin impinging jet in cross-flow problems. Complex geometry was handled by using multiblock grids. The LES solutions gave reasonable agreement with experiment and were

Figure 1. Vertical landing aircraft flow features.

a significant improvement over RANS solutions. For example, the LES predicted the experimental fountain strength, which was under-predicted by the RANS, and matched the experimental ground vortex strength. The interaction of the wall jets to form an unsteady fountain that sometimes reached the intake was observed. There was a disparity in time scales between that needed to resolve the smallest eddies and that for slow cross-flow to travel through the domain, resulting in the computation needing a large number of time steps to achieve statistically meaningful results; this was particularly noticeable in the poor quality of the mean LES field near the intake.

Industrial applications, however, frequently arise in combination with complex geometry. Tetrahedral meshes can be easily generated for complex geometry, but solutions on these meshes tend to show more numerical diffusion than an equivalent hexahedral mesh. The approach taken in the current work is to utilise an unstructured hexahedral mesh in regions where turbulent fluctuations are important, combined with a tetrahedral mesh to provide geometrical flexibility. Previous to this work, a low Mach number fully developed turbulent pipe flow and a Mach number 0.9 round jet flow have been simulated in order to test the compressible solver LES suitability for wall-bounded and free shear flows⁽¹⁶⁾. Good agreement in terms of mean and turbulence quantities were obtained compared to previous work.

The current work complements the study of Richardson, *et al*⁽¹⁷⁾ which has demonstrated moving mesh unstructured RANS solutions for a descending aircraft.

The aim of this study is to assess the suitability of LES for the prediction of multiple jet impingement problems relevant to vertical landing aircraft, with the intention in the future to apply the unstructured LES methodology to geometrically complex aircraft configurations.

2.0 METHODOLOGY

2.1 Solution algorithm

The starting point for this work is the Rolls-Royce CFD code Hydra⁽¹⁸⁾ which is an unstructured, mixed element, compressible, density-based Reynolds Averaged Navier-Stokes solver. The discretisation was improved so as to avoid excessive dissipation of resolved eddies and subgrid scale models incorporated. The important features are summarised below, and further details of the discretisation and testing on simpler LES flow problems can be found in Tristante *et al*⁽¹⁶⁾.

2.2 Governing equations

Employing Cartesian tensor notation and the conservative variables $(\rho, \rho u_i, E)$, the governing time dependent equations in terms of spatially filtered, Favre-averaged compressible N -S equations can be expressed as;

$$\frac{\partial}{\partial t} \iiint_{\Gamma} Q dV + \iint_{\Gamma} F(Q) \bullet n dS + \iint_{\Gamma} F(Q) \bullet n dS = 0, \quad \dots (1)$$

where;

$$Q = \begin{bmatrix} \bar{\rho} \\ \bar{\rho} \tilde{u} \\ \bar{\rho} \tilde{v} \\ \bar{\rho} \tilde{w} \\ \bar{E} \end{bmatrix}, F(Q) \bullet n = \begin{bmatrix} \bar{\rho} \tilde{U}_n \\ \bar{\rho} \tilde{U}_n \tilde{u} + n_x \bar{p} \\ \bar{\rho} \tilde{U}_n \tilde{v} + n_y \bar{p} \\ \bar{\rho} \tilde{U}_n \tilde{w} + n_z \bar{p} \\ \bar{U}_n (\bar{E} + \bar{p}) \end{bmatrix},$$

and $G(Q)$ contains viscous and conduction flux terms. The finite volume discretisation provides an implicit filter for the large eddies.

while $\bar{\cdot}$ denotes unweighted filtered variables and $\tilde{\cdot}$ density weighted filtered variables. The spatial filter size is computed at every node from the control volume surrounding the node. The finite volumes are created from the median-dual of the original unstructured mesh which may contain tetrahedra, hexahedra, pyramids and prisms.

2.3 Discretisation

For an edge ij that connects nodes i and j , the flux is computed using a second-order accurate scheme of Moinier⁽¹⁹⁾

$$F_{ij} = \frac{1}{2} [F(Q_i) + F(Q_j) - \text{smoothing}], \quad \dots (2)$$

The smoothing term is defined as⁽¹⁸⁾,

$$\text{smoothing} = |A_{ij}| \varepsilon_1 (L_j^{lp}(Q) - L_i^{lp}(Q)), \quad \dots (3)$$

where L is the pseudo-Laplacian and

$$|A_{ij}| = \partial F / \partial Q \quad \dots (4)$$

For LES it is essential that the smoothing term should be kept as small as possible so as to avoid unphysical dissipation of the resolved eddies. This is achieved by the use the sensor function of Ducros *et al*⁽²⁰⁾ to control ε_1 , based upon the vorticity Ω and divergence $\nabla \bullet \mathbf{u}$.

$$\varepsilon_1 = \max \left(\varepsilon_2, \varepsilon_3 \frac{(\nabla \bullet \mathbf{u})^2}{(\nabla \bullet \mathbf{u})^2 + \Omega^2} \right), \quad \dots (5)$$

where ε_2 and ε_3 are user defined parameters. The sensor increases the level of smoothing for regions of high divergence and reduces it to a base level of ε_2 for regions of high vorticity. In some cases, particularly at jet impingement, unphysical oscillations were observed in the near wall region and the smoothing was locally increased in the cells closest to the wall to damp the oscillations.

Temporal discretisation used a third order accurate, three-stage Runge-Kutta algorithm⁽²¹⁾.

2.4 Sub grid scale model

The standard Smagorinsky SGS model defines the subgrid scale viscosity μ_t as

$$\mu_t = C_s^2 \rho \Delta^2 \sqrt{2 S_{ij} S_{ij}}, \quad \dots (6)$$

where the strain rate is

$$S_{ij} = \frac{1}{2} \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right). \quad \dots (7)$$

and Δ is the filter width.

For the correct prediction of a laminar flow or the viscous sublayer of a turbulent flow, the SGS model should tend to zero in these regions. This is not true for a fixed-coefficient Smagorinsky model, in particular in the near wall region where the S_{ij} term becomes large. An improvement on the basic Smagorinsky model is the wall-adapting local eddy-viscosity (WALE) model, proposed by Nicoud and Ducros⁽²²⁾ for LES in complex geometries. This model is based on the square of the velocity gradient tensor and accounts for the effects of both the strain and the rotation rate of the smallest resolved turbulent fluctuations, and it also produces the correct scaling at the wall ($\nu_t = o(y^3)$) without the explicit use of the local wall distance.

Figure 2. Twin-jet: configuration.

Figure 3. Twin-jet: grid detail in x-y plane.

Figure 4. Twin-jet: grid detail in y-z plane.

The WALE model defines the subgrid scale viscosity as;

$$\mu_t = C_w^2 \rho \Delta^2 \frac{OP1}{OP2}, \quad \dots (8)$$

where

$$OP1 = (S_{ij}^d S_{ij}^d)^{3/2}, \quad \dots (9)$$

$$OP2 = (S_{ij} S_{ij})^{5/2} + (S_{ij}^d S_{ij}^d)^{5/4}, \quad \dots (10)$$

$$S_{ij}^d = \frac{1}{2} \left(\frac{\partial u_i}{\partial x_k} \frac{\partial u_k}{\partial x_j} + \frac{\partial u_j}{\partial x_k} \frac{\partial u_k}{\partial x_i} \right) - \frac{1}{3} \delta_{ij} \frac{\partial u_k}{\partial x_k} \frac{\partial u_k}{\partial x_k}, \quad \dots (11)$$

This model has been demonstrated to be independent of near-wall distance with natural recovery of sublayer flow conditions; zero subgrid scale viscosity is recovered at the wall without an explicit damping function. C_w is a model constant and for the calculations presented here, C_w has been set to a value of 0.5 as recommended by Nicoud and Ducros⁽²²⁾. The filter width, Δ , is determined from the cube root of the control volume,

$$\Delta = \Gamma^{1/3} \quad \dots (12)$$

2.5 Computations

All calculations have been carried out in parallel. The domain is partitioned at run-time according to the number of processors available and the sub-domains communicate using MPI. Initial testing of the calculations was undertaken on a local AMD Opteron cluster, and then transferred to the UK HPCx facility for the final longer calculations. Typically a single 'frame' was used with 32 IBM Power5 processors and required approximately four seconds of 'wall time' per computational time step. Consequently, the total wall time for a twin impinging jet calculation was approximately ten days. Taking into account the need to queue jobs, the total turnaround time to achieve a solution was approximately four weeks.

3.0 RESULTS

3.1 Twin-jet in cross-flow

3.1.1 Configuration

Barata *et al*⁽⁶⁾ carried out experiments in a water tunnel with a jet to cross-flow ratio of $W_j/U_c = 30$, an impingement height of $5D_j$ and a jet centreline spacing of $5D_j$ – these parameters being representative of typical vertical landing aircraft. The Reynolds number based upon jet velocity and diameter was 105,000. The configuration of the computational domain is shown in Fig. 2.

3.1.2 Mesh and boundary conditions

The computational grid was generated using purely hexahedral elements with $153 \times 203 \times 67$ points in the cross-flow, jet and transverse directions, giving a total of two million nodes (Figs 3 and 4). It should be noted that the solver treats this as an arbitrary unstructured mesh, although it has been generated as a single curvilinear structured block. The vertical spacing was $0.004D_j$ in the near wall region and $0.05D_j$ at the jet exit. In the jet regions, the cross-flow and transverse spacing was $0.043D_j$. This spacing was based upon previous

Figure 5. Turbulence structures: $Q_c = 0.5$.

Figure 6. Velocity magnitude contours.

LES experience with this problem using a structured solver⁽¹⁵⁾ as well as the calculation of a free jet using the current method⁽¹⁶⁾. Examination of these earlier structured solutions did indicate that mesh resolution in the impingement zone was important and so, in comparison to the earlier meshes, the wall normal spacing was halved in this area. This also led to a better resolution of the cross-flow boundary-layer. This boundary-layer thickness was approximately $0.1D_j$ when it reached the jet impingement zone, and was resolved by more than 20 points.

The flow upstream of the nozzle exit plane is not modelled. The calculation was initialised with the cross-flow velocity throughout the domain, and the jets were impulsively started. The jet velocity is fixed at the required value and no random perturbations are applied: the free shear layer of the jet rapidly transitions to turbulence without needing any extra forcing. Similarly, the cross-flow velocity is simply fixed to the required value.

The time step was initially defined to resolve the smallest resolved eddies in the jet shear layer and impingement zone. Non-dimensionalised by a time scale based upon jet diameter and velocity, the time step is 0.001. It was found that the compressible nature of the solver could result in the maximum allowable timestep for stability being considerably smaller than that needed to resolve the smallest eddies.

Experimentation with increasing the jet Mach number from 0.05 to 0.3 (while retaining the correct Reynolds number and jet to cross-flow ratio) allowed a larger time step to be used, without affecting the mean solution. The time step is still considerably smaller than the earlier pressure-solver results⁽¹⁵⁾ and results in a temporally over-resolved solution.

The simulation time to gather statistics corresponds to approximately one flow through time based upon cross-flow velocity, although the impingement and fountain region achieve a statistically steady state in this time because of the much shorter time scales related to the jet velocity and impingement height. This is a significant issue in these types of problems, where the computational time step is determined by the high speed flow in small cells in the jet, while the time scale of the vortex and cross-flow is determined by a much smaller velocity and a very large domain.

3.1.3 Simulations

An instantaneous eddy structure is visualised in Fig. 5 as an isosurface of the positive invariant of velocity divergence⁽²³⁾ ($Q_c = 0.5$). Instabilities in the jet shear layer cause large-eddy structures

(a) Mean velocities.

(b) Means streamwise velocity in symmetry plane.

Figure 7.

(vortex rings) to be visible prior to one jet diameter downstream of the nozzle. In contrast to a free jet simulation which can often have an excessively long 'laminar like' initial shear layer, the turbulence from the fountain induces instability in the jet shear very rapidly leading to a fully turbulent flow. The turbulent structures are larger than those observed in a free jet by Krothapalli *et al.*⁽⁸⁾. These eddies feed into the fountain structure, which can be seen to be highly asymmetric. Observation of animations of the CFD prediction show the fountain to be 'flapping' between the jets, which in turn enhances the unsteadiness of the jet shear layer adjacent to the fountain. There are three stagnation points close to ground wall, which form the 'flapping' bubbles and fountain base (see Fig. 6). Figure 7 (a) shows the mean vertical velocity across the jets at three different locations from the impingement plane, $z/D_j = 0.25, 1, 2$. These are compared to the experimental measurements of Barata *et al.*⁽⁶⁾ and earlier CFD predictions of Page *et al.*⁽¹⁵⁾ that used a pressure based solver on a structured grid. All three sets of data are in agreement at $z/D_j = 0.25$, while the present results have a weaker fountain strength at $z/D_j = 1$ and $z/D_j = 2$ than both experiment and the earlier predictions. In contrast, the present results obtain the correct experimental width of the jet shear layer, while the earlier predictions have a somewhat wide jet. Fig. 7(b) shows the streamwise velocity in the streamwise symmetry plane at the above three locations. The forward and backward flow penetrations agree very well with experiment. Turbulent normal stresses for the same locations are presented in Fig. 8(a,b), which give the same trend and magnitude as experiment. Since the results of Page *et al.*⁽¹⁵⁾ show increased

(a) Streamwise normal stress.

(b) Vertical normal stress.

Figure 8. Twin jet, RMS velocity profile.

spreading of the jet shear layers (Fig. 7), this results in the peaks in normal stress being too far apart at all locations. The present results are in better agreement for the peak location because of the improved jet shear layer growth, but tend to under-predict the streamwise normal stress and over-predict the vertical normal stress. The experiments show higher levels of fluctuation on the inboard shear layer as the fountain flow is feeding turbulence into the early development of the shear layer. In general, the results of Page *et al.*⁽¹⁵⁾ show elevated inboard stresses, whereas the present results do not have this asymmetry. Examination of the fountain velocity (Fig. 7) show that the present results have a significant decay in fountain strength at $z/D_j = 2$, so leading to less fluid being entrained into the early part of the inboard shear layer and consequently elevated normal stresses are not observed. As the mesh resolution is very similar for the two sets of calculations, it would suggest that there is an increase in numerical dissipation in the unstructured results that may require increased mesh resolution to avoid.

It should be noted that the LES solution has not been additionally averaged across the two halves of the domain so as to give perfectly symmetric mean velocity and normal stress profiles – the small asymmetry is an indication that further samples should be included to improve the normal stress prediction. Overall there is good agreement with experiment.

Figure 9(a,b) show energy spectra in the jet shear layers at the previous three vertical stations. The same information is shown in both graphs, but the semi-log profile in (a) is used to emphasise the strength and location of the dominant frequency. The dominant

(a)

(a) Energy spectra in fountain.

(b)

(b) 0.25Dj from wall

Figure 9. Energy spectra in jet shear layers.

Figure 10.

frequency is at a Strouhal number (based on jet velocity and diameter) of 0.1004 ($f_{jet} = 1,550\text{Hz}$) and the strongest fluctuations are observed at the location closest to the ground plane. This Strouhal number is somewhat lower than that observed in free jets; for example Bogey and Bailly⁽²⁴⁾ show the peak in the spectral power density at Strouhal numbers of 0.4 to 0.9 for an LES of a subsonic free jet at a similar Reynolds number. It appears that the most energetic fluctuations are due to unsteadiness introduced by the impingement process, rather than instabilities in the free shear layer. These are of largest magnitude near the ground plane, but the fountain flow feeds into the inner part of the jet shear layer, forcing the jet at this lower Strouhal number. Time histories have also been captured on the symmetry plane to show the fountain behaviour – this is shown as spectra in Fig. 10. The greatest fluctuations occur at the station closest to the ground, and this station is shown in more detail in Fig. 10(b). To compare to the jet shear layer behaviour, the Strouhal numbers are normalised by the dominant jet shear layer frequency (1,550Hz) and the jet shear layer energy spectra at the same height is included in the plot. First, there is considerably more energy in the fountain fluctuations, and secondly the Strouhal number of the dominant fountain fluctuations is a factor of 1.1 greater than the jet shear layer Strouhal number. Also included in the figure are spectra for locations on the symmetry plane, one jet diameter ahead of the fountain centre ($X+$) and one jet diameter downstream ($X-$). The forward location has the same amplitude and frequency as the fountain, while the rearward location has a similar amplitude and frequency to the jet shear layer.

3.2 Twin-jet with intake in cross-flow

3.2.1 Configuration

While the simple twin-jet problem contains most of the important flow features of a vertical landing aircraft, it is limited in terms of understanding hot gas ingestion as there is no mechanism for fluid to be re-ingested. The second configuration includes a circular intake with a mass flow equal to that from the two jets (see Fig. 11). The location of the intake in relation to the jet nozzles is similar to a Harrier aircraft. Feed pipes lead from the top wall of the water tunnel to the jet nozzles and a return pipe removes the intake flow. This water-rig configuration was studied experimentally using LDA by Behrouzi and McGuirk⁽⁷⁾ specifically to provide validation data for CFD. Three jet to cross-flow velocity ratios, $R = 18, 24, 35$ were studied experimentally. It was found that the $R = 18$ case had insignificant ingestion, while the $R = 35$ case had practically continuous ingestion. The $R = 24$ case together with an impingement height of $7D_j$ gave intermittent ingestion and is chosen as a difficult case for the LES methodology. Since the LES calculations are compressible, the cross-flow velocity was adjusted to ensure that the mass-flow ratio $(\rho_j W_j)/(\rho_c U_c)$ was set to 24, where ρ_c and U_c represent the cross-flow density and velocity. The mass-flow ratio was matched in preference to the velocity or momentum ratio as this is typically carried out in the similar gas turbine combustor port flow problem. The jet Reynolds number based on jet velocity and diameter is $Re_j = 40,000$.

Figure 11. Twin jet with intake: geometry.

(a) Nozzle exit.

(b) $2D_j$ from impingement plane.

Figure 13. Mean vertical velocity component.

(a) Top view

Figure 12. Twin jet with intake: unstructured mesh.

3.2.2 Mesh and boundary conditions

The complexity of the geometry (and in particular the region around the feed pipes) means that an unstructured mesh generation approach is essential. For the jet impingement, ground vortex and fountain regions, the mesh contains hexahedral elements with a high mesh density (see Fig. 12). ‘O-mesh’ features are used to resolve the circular jets and intake. Away from this region the mesh uses tetrahedral elements to mesh the upper part of the feed pipes and expand towards the upper and side walls. The more sophisticated meshing topology allowed a mesh to be generated inside the nozzles resolving three jet diameters upstream of the nozzle exit plane. Although there is no resolved LES in the nozzle, there were some observations in the earlier test case that starting the solution at the nozzle exit plane could result in some spurious reflections from this boundary condition. The mesh was designed to have a similar resolution in the jet and impingement

(a) $y/D_j = 0.0$ (a) Streamwise velocity component at $z/D_j = 0.25$ (b) $x/D_j = 9.5$. (b) Vertical velocity component at $x/D_j = 0.0$ Figure 14. Mean streamwise velocity component at $x/D_j = -7$.

Figure 15. Fountain symmetry plane at

regions to that of the previous case and has a similar total number of nodes (1.8 million). The hybrid mesh, however, does increase the total work to achieve a solution as the number of edges is 6.5 million. Again, there are no perturbations applied to the jet inflow or cross-flow boundary conditions and the time-step, non-dimensionalised by jet velocity and nozzle diameter, was 0.001.

3.2.3 Simulations

Figure 13 shows profiles of mean velocity across the jet (i.e. in a plane connecting the jet axes) at a station immediately downstream of the nozzle and at $2D_j$ from the impingement plane. The simulations have predicted the correct spreading of the jet and the correct fountain strength and width.

Experimental data were available in a vertical plane at $x/D_j = -7$, this being just ahead of the intake. The simulations show an

underprediction of the increased velocity due to suction at $z/D_j = 10$ (Fig. 14(a)) and a strange behaviour of the profile nearer to the ground. Due to the much longer time scales in this region, it is possible that insufficient samples have been taken to give an accurate mean solution. It is also possible that the mesh expansion in this region was too great giving a poorly resolved LES. Profiles are also shown the domain at $z/D_j = 9.5$ in Fig. 14(b), this shows that the simulations are producing the 'correct' mean solution due to the intake, but that the local mean cross-flow velocity is too low. The blockage of the feedpipes and jet flow causes a local acceleration of the cross-flow which is more pronounced in the experiment.

Figure 15 shows results on the symmetry plane. The streamwise velocity component is shown in Fig. 15(a) at $0.25D_j$ from the impingement plane. In the region upstream of $x/D_j = -10$ the predictions are poor due to the grid size expansion giving a poorly

(a) Upwash fountain, contours of velocity magnitude.

(b) Swirl in intake

Figure 16. Unsteady flow visualisation near intake.

resolved simulation. Similarly, the oscillations in the prediction at $z/D_j = -6$ may also be a grid quality issue (although, curiously, there is a single experimental data point slightly upstream with a similar behaviour). Nevertheless, there is good agreement with experiment in the region $-4 < x/D_j < 4$ at this height from the ground. The upwash fountain decay rate is an important parameter and this is shown in Fig. 15(b) as the variation of the vertical component of velocity with height. Apart from an under-prediction between one and two diameters from impingement this is in excellent agreement with experiment.

Of importance in the design context is the flow unsteadiness near the intake face. Instantaneous flow solutions are visualised in Fig. 16. The forward flowing fountain is strongest near the impingement plane (Fig. 16(a)) and flaps around significantly, often reaching up to the intake. When the rotational flow is accelerated into the intake, the cross-section of the vortex reduces in area with a consequent increase in rotational speed. The swirl shown in Fig. 16(b) would clearly be detrimental to the performance and stability of a gas-turbine compressor.

The experimental study visualised the flow in the vicinity of the intake using a double-pulsed laser sheet. Example pictures are

(a)

(b)

Figure 17. Experimental flow visualization.

shown in Fig. 17 and turbulent structures are observed predominantly near the lower half of the intake. An instantaneous computational solution is visualised by computing pseudo-streamlines on a frozen flow-field (Fig. 18); these are not true streamlines as the flow is time varying. While it is difficult to compare instantaneous flow visualisations from experiment and LES, there is clearly a highly unsteady fountain flow that enters the intake from below.

Although the experiments were isothermal, the computational simulation was set up with a small increase in temperature for the jets. This was sufficiently small that the small change in density would not affect the flow, but would allow tracking of jet fluid into the intake. The scalar temperature field is calculated to be in the range zero to unity

$$T_{sc} = (T - T_c)/(T_j - T_c) \quad \dots (13)$$

where subscripts j and c refer to the jet and cross-flow respectively. Contour plots are shown in Fig. 19 and the predicted temperature increase in the intake is approximately 50% of the excess jet temperature.

(a) Symmetry plane.

(a) Symmetry plane.

(b) Intake detail.

(b) Front view

Figure 18. Computational flow visualization, instantaneous 'streamlines'.

Figure 19. Scalar temperature contours.

4.0 CONCLUSIONS AND FUTURE WORK

Large-eddy simulations of both a simple twin impinging jet flow and a more complicated configuration including an intake system are in good agreement with experiment for the mean flow and in qualitative agreement with intake flow visualisation. The dominant frequency of the fluctuations was at a Strouhal number of 0.1, somewhat lower than normally observed in free jets. The dominant unsteadiness is due to the impingement process and the flapping motion of the fountain.

The structured mesh approach allowed the resolution of the jet feed and intake system while retaining hexahedral elements in the important unsteady flow-field region. Computational resources for these calculations were relatively modest as even the more complex geometry was resolved using 1.8 million nodes.

These results show considerable promise for the utility of large-eddy simulation when applied to multiple jet impingement with realistic aircraft configurations.

ACKNOWLEDGEMENTS

Funding for this project was from the UK Engineering and Physical Sciences Research Council (EPSRC) grant GR/S21021/01. The computer time was provided through the UK Applied Aerodynamics Consortium (UKAAC) under EPSRC grant GR/S91130/01.

The authors would like to thank Dr Giles Richardson at the University of Cambridge for the generation of the unstructured mesh used in the twin jet with intake case, and Dr Nick Nills at the University of Surrey for guidance on carrying out these calculations in parallel.

REFERENCES

1. CURTIS, P. A review of the status of ground effect/environment technologies, No 2002-5985, November 2002, 2002 Biennial International Powered Lift Conference and Exhibit, November 2002, Williamsburg, Virginia, USA.
2. SEDDON, J. and GOLDSMITH, E. *Intake Aerodynamics*, Blackwell Science Ltd, 1999, 2nd ed.
3. PAGE, G.J., MCGUIRK, J.J., HARPER, L.R. and PENROSE, C.J. Application of computational fluid dynamics to hot gas jet engine modelling, August

- 1998, Proceedings of the Royal Aeronautical Society International Powered Lift Conference, London, UK.
4. CHADERJIAN, N.M., PANDYA, S.A., AHMED, J.U. and MURMAN, S.M. progress toward generation of a Navier-Stokes database for a Harrier in ground effect, No. 2002-5966, November 2002, 2002 Biennial International Powered Lift Conference and Exhibit, Williamsburg, Virginia, USA.
 5. PANDYA, S.A., MURMAN, S.M. and SANKARAN, V. Unsteady computations of a jet in crossflow with ground effect, No 2003-3890, June 2003, 33rd AIAA Fluid Dynamics Conference and Exhibit, Orlando, Florida, USA.
 6. BARATA, J., DURAO, D., HEITOR, M. and MCGUIRK, J. Impingement of single and twin turbulent jets through a crossflow, *AIAA J*, April 1991, **29**, (4), pp 595-602.
 7. BEHROUZI, P. and MCGUIRK, J. Experimental data for CFD validation of the intake ingestion process in STOVl aircraft, *J Flow Turbulence and Combustion*, 2000, **64**, pp 233-251.
 8. KROTHAPALLI, A., RAJKUPERAN, E., ALVI, F. and LOURENCO, L. Flow field and noise characteristics of A supersonic impinging jet, *J Fluid Mechanics*, 1999, **392**, pp 155-181.
 9. CABRITA, P., SADDINGTON, A. and KNOWLES, K. Unsteady features of twin-jet STOVl ground effects, No 2002-6014, 2002 Biennial International Powered Lift Conference and Exhibit, November 2002, Williamsburg, Virginia, USA.
 10. ALKISLAR, M., KROTHAPALLI, A., CHOUTAPALLI, I. and LOURENCO, L. Structure of supersonic twin jets, *AIAA J*, November 2005, **43**, (11), pp 2309-2318.
 11. LOU, H., ALVI, F.S. and SHIH, C. Active and passive control of supersonic impinging jets, *AIAA J*, January 2006, **44**, (11), pp 58-66.
 12. CHILDS, R. and NIXON, D. Unsteady three-dimensional simulations of a VTOL upwash fountain, No 86-0202, AIAA 24th Aerospace Sciences Meeting, 6-9 January 1986, Reno, Nevada, USA.
 13. CHILDS, R., NIXON, D., KUHN, G. and PERKINS, S. Further studies of impinging jet phenomena, No 87-0017, January 1987, AIAA 25th Aerospace Sciences Meeting, 12-15 January 1987, Reno, Nevada, USA.
 14. RIZK, M. and MENON, S. An investigation of excitation effects on a row of impinging jets using large-eddy simulations, No 88-0043, January 1988, AIAA 26th Aerospace Sciences Meeting, 11-14 January, 1988, Reno, Nevada, USA.
 15. PAGE, G.J., LI, Q. and MCGUIRK, J.J. LES of impinging jet flows relevant to vertical landing aircraft, No 2005-5226, 2005, 23rd AIAA Applied Aerodynamics Conference, 6-9 June 2005, Toronto, Ontario, Canada.
 16. TRISTANTO, I.H., LI, Q., PAGE, G.J. and MCGUIRK, J.J. On the effect of convective flux formation for LES of compressible flows using hybrid unstructured meshes, No 2006-54940, 2006, 36TH AIAA Fluid Dynamics Conference, 5-8 June 2006, San Francisco, California, USA.
 17. RICHARDSON, G., DAWES, W. and SAVILL, M. Unsteady, moving mesh CFD simulation applied to Harrier VSTOL aircraft for hot-gas ingestion control analysis, *Aeronaut J*, March 2007, **111**, (1115).
 18. CRUMPTON, P.I., MOINIER, P., and GILES, M.G. An unstructured algorithm for high Reynolds number flows on highly stretched grids, 1997, Tenth International Conference on Numerical Methods for Laminar and Turbulent Flow.
 19. MOINIER, P. Algorithm Developments for an Unstructured Viscous Flow Solver, PhD thesis, University of Oxford, 1998.
 20. DUCROS, F., FERRAND, V., NICOUD, F., WEBER, C., DARRACQ, D., GACHERIU, C. and POINSOT, T. Large-eddy simulation of the shock/turbulence interaction, *J Computational Physics*, 1999, **152**, pp 517-549.
 21. GOTTLIEB, S. and SHU, C.W. Total variation diminishing Runge-Kutta Schemes, Tech Rep 96-50, ICASE, 1996.
 22. NICOUD, F. and DUCROS, F. Subgrid-scale stress modelling based on the square of the velocity gradient tensor, flow, *Turbulence and Combustion*, 1999, **62**, pp 183-200.
 23. JEONG, J. and HUSSEIN, F. On the identification of a vortex, *J Fluid Mechanics*, 1995, **285**, p p 69-94.
 24. BOGEY, C. and BAILLY, C. Effects of inflow conditions and forcing on subsonic jet flow and noise, *AIAA J*, 2005, **4**, pp 1000-1007.