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.

Computational fluid dynamics prediction of intake ingestion relevant to short take-off and vertical landing aircraft

PLEASE CITE THE PUBLISHED VERSION

PUBLISHER

Professional Engineering Publishing / © IMECHE

VERSION

VoR (Version of Record)

LICENCE

CC BY-NC-ND 4.0

REPOSITORY RECORD

Behrouzi, P., and James J. McGuirk. 2009. "Computational Fluid Dynamics Prediction of Intake Ingestion Relevant to Short Take-off and Vertical Landing Aircraft". figshare. <https://hdl.handle.net/2134/4772>.

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/>

Computational fluid dynamics prediction of intake ingestion relevant to short take-off and vertical landing aircraft

P Behrouzi* and J J McGuirk

Department of Aeronautical and Automotive Engineering, Loughborough University, Loughborough, Leicestershire, UK

Abstract: Intake ingestion can cause several major problems (e.g. compressor surge and stall) for short take-off and vertical landing (STOVL) aircraft operating in ground effect. Numerical predictions of the flowfield associated with a generic twin-jet plus intake model operating under ingestion flow conditions are reported using computational fluid dynamics (CFD) techniques. The results have been compared with laser Doppler anemometry (LDA) validation measurements taken in a specially designed test case configuration. The $k-\varepsilon$ turbulence model and both first-order and second-order (QUICK) convection discretization schemes were employed. Fine meshes and second-order accurate discretization were found essential to produce solutions close to grid independence. A reasonable prediction of the general flow pattern has been achieved. Several features of the mean velocity field were close to the experimental results; however, the $k-\varepsilon$ model was shown to produce significant errors in the prediction of the forward penetration distance of the ground sheet flow and in the shape of velocity profiles and turbulence levels near to the intake.

Keywords: computational fluid dynamics, STOVL aircraft, intake ingestion

NOTATION

d_h	water tunnel test section hydraulic diameter
d_j	jet exit diameter
n_x	number of grid nodes in the x direction
n_y	number of grid nodes in the y direction
n_z	number of grid nodes in the z direction
R	velocity ratio = V_j/W_c
U	transverse velocity component
V	vertical velocity component
V_j	jet velocity
W	longitudinal velocity component
W_c	crossflow velocity
x	transverse coordinate
y	vertical coordinate
z	longitudinal coordinate
ν	dynamic viscosity of water

1 INTRODUCTION

A short take-off and vertical landing (STOVL) aircraft, such as the British Aerospace Harrier, in its landing phase of flight operation, creates a complex three-dimensional flow between lift jet streams, the airframe surface and the ground. The flowfield consists of jet plumes, ground impingement, fountain up-wash flow, wall jets with ambient entrainment, headwind (or crossflow) and ground vortex flows. These flows drastically change the aerodynamic forces on the aircraft, as well as create serious design and operational problems such as hot-gas ingestion, suck-down forces (lift loss) and jet-induced lift. The ingestion of exhaust gases into the engine intake, as sketched in Fig. 1, leads to a time-dependent rise in intake temperature and spatial non-uniformities in pressure and velocity in the intake flow. The unsteadiness, disturbance of intake airflow quality and rise in temperature can cause serious problems for engine performance, including thrust reduction and compressor surge and stall. Although the temperature rise is ultimately the most significant parameter, density variations (due to pressure or temperature changes) probably exert only a secondary influence on the flow pattern in the immediate intake vicinity. In addition, the interacting fluid mechanics of the various flow features illustrated in Fig. 1 is so complex that it is of interest to concentrate initially on the aerodynamic phenomena leading to intake

The MS was received on 27 November 1997 and was accepted after revision for publication on 21 May 1999.

**Corresponding author: Department of Aeronautical and Automotive Engineering, Loughborough University, Loughborough, Leicestershire LE11 3TU, UK.*

Fig. 1 Major aerodynamic features influencing intake ingestion for STOVL aircraft in ground effect

ingestion in the absence of density variation. This is the approach adopted in the present paper.

During the past 20–30 years, the flow characteristics associated with the ingestion process have been studied extensively using both small- and large-scale model testing. Most of the ingestion studies have been made on specific configurations. There have been very few general research investigations where configuration variables were systematically varied. Preliminary design methods for predicting ingestion effects usually depend heavily on empirical correlations [1]. Information available in the literature does indicate that ingestion characteristics are configuration dependent, but there is still a need to understand the basic flow processes. Considerable effort has gone into analysing and predicting the overall flowfield in the past (e.g. see references [1] to [8]). Ground-effect studies are summarized by Stewart and Kuhn [6], who give additional insight into the mechanism of exhaust gas ingestion, particularly at low forward speeds. Although the basic flow mechanisms that dominate ground effect flows around STOVL aircraft are known, details of these mechanisms are not adequately understood.

The application of computational fluid dynamics (CFD) techniques to the ground-effect ingestion problem has increased during recent years. As a logical step towards the analysis of a complete STOVL aircraft, flowfields associated with geometrically simplified models such as single, double or multiple jet impingement, with or without crossflow, have been studied. These generic flowfields isolate the physics of multiple jets, jet/ground plane or jet/headwind interaction without the addition of other complications such as complex aircraft geometry. In the last decade extensive experimental and computational studies have been conducted to study the fluid mechanics of single and multiple impinging jets with and without crossflow. References [9] to [29] have provided general reviews of current understanding of attempts to compute these flowfields.

Numerical calculations of impinging jets through a crossflow for high-velocity ratios and small impingement heights have been reported by Barata *et al.* [13]. Behrouzi and McGuirk [14] and Barata *et al.* [15] have also provided numerical calculations for a single jet configuration, including an examination of the capabilities of the computational method based on detailed comparison with

measurements. Jones and McGuirk [16] presented early calculations of non-impinging jets in crossflow using the standard two-equation $k-\epsilon$ model. However, the use of first-order hybrid differencing for convection and very coarse meshes made any statements on turbulence model accuracy difficult due to the intrusion of numerical diffusion. Several solutions of the Reynolds averaged Navier–Stokes (RANS) equations have already been computed for both single [17–23] and multiple jets in crossflow [24–26] remote from a ground plane, but these studies are perhaps more relevant to the transition phase of STOVL aircraft operation rather than to in-ground-effect phenomena. Childs and Nixon [27] presented calculations relevant to ground-effect flow using the $k-\epsilon$ model and confirmed the gross features of the flow, but little comparison was given with experimental data to enable a quantitative judgement of the calculations. Claus [26] has addressed the question of false diffusion in three-dimensional jet-in-crossflow calculations and found it is necessary to adopt second-order QUICK discretization for the convective terms in order to obtain numerically accurate solutions. The $k-\epsilon$ turbulence model with the QUICK numerical scheme was also successfully employed by Marquis [28], who concluded that the turbulent structure of the flow was independent of the velocity ratio ($R = \text{jet velocity/crossflow velocity}$) as long as this was relatively high (of order 30).

More recently, several higher-resolution computations, each employing at least 100 000 grid points to discretize the three-dimensional domain, have been reported. A detailed picture of the jet plume and a discussion of the vorticity dynamics for a turbulent jet with velocity ratios between 2 and 8 has been given by Sykes *et al.* [20]. However, since their primary motivation was the behaviour of the jet remote from the wall, they did not attempt to resolve details of the wake region. Claus and Vanka [29] presented a series of calculations which attempted to study the grid-independent behaviour of the $k-\epsilon$ model for jet-in-crossflow problems. Up to 2.5 million grid nodes were used; their study indicated that, for calculations employing a first-order scheme, grid independence is probably impossible to achieve, even with grids as fine as this.

In spite of the difficulties described above, the availability and capability of current three-dimensional CFD codes has encouraged their application as a power-

ful tool in analysing the hot gas environment around a STOVL aircraft operating in ground proximity. Van Overbeke and Holdeman [30] have reported a CFD study of a typical STOVL aircraft configuration to identify key features of the flowfield and to demonstrate and assess the capability of current CFD codes to calculate the temperature of the gases ingested at the engine inlet as a function of flow and geometric conditions. In another study [31], the same authors included a complete fuselage, headwind and multiple impinging jets in their numerical simulations. By varying the crossflow and the impingement height, they were able to alter the ingestion pattern and quantify the effect of headwind. The hot gas ingestion problem for a STOVL aircraft was modelled as multiple jets in crossflow with inlet suction by Fricker *et al.* [32]. A geometrically simple model of the aircraft with four choked jets was studied at various heights, headwind speeds and lift jet splay angles in a modest parametric study. Tafti and Vanka [33] used a multigrid calculation procedure for three-dimensional flows for similar ingestion studies. Calculations with a simulated fuselage and twin exhaust jets were made for two ground positions and two headwind strengths. The global features of the flowfield such as the ground plane temperature distributions and the formation of the ground vortex were reproduced as expected, but little detailed validation of the flowfield was carried out.

As part of a continuing programme of experimental and computational work aimed at identifying and quantifying the important mechanisms of intake ingestion flows, the present paper reports on CFD studies similar to those reported in references [31] to [33]. The approach adopted differs in that the CFD predictions are carried out for a specially designed generic twin jet/intake configuration to allow specific emphasis to be placed on the complex aerodynamic flowfields encountered in intake ingestion problems. Further, the results of the

present numerical studies are compared with high-quality laser Doppler anemometry (LDA) measurements taken in the same generic configuration by Behrouzi and McGuirk [34].

2 NUMERICAL APPROACH

The predictions in this paper have been performed with a modified version of the finite volume, pressure-correction-based code used in reference [15] for solution of the RANS equations for an isolated jet impingement problem. The modification consisted primarily of the introduction of an automatic procedure based on labelling of the finite volume cells to allow blockages internal to the solution domain to be created. This was essential for the introduction of the intake duct and jet discharge pipes within the water tunnel test section. This also allowed the measured boundary conditions at jet exit and intake entrance to be fixed. All calculations have adopted the high Reynolds number $k-\epsilon$ eddy viscosity turbulence model, together with log-law-based wall functions on solid surfaces [35]. In terms of convection discretization, two practices have been used: standard first-order (hybrid) differencing and second-order (QUICK) differencing [36] to allow assessment of numerical accuracy.

For these initial studies, a Cartesian mesh (grid size given in the next section) was used throughout the whole flowfield and the jet nozzle and intake pipe ductwork shown in Fig. 2 were treated as blockages internal to the grid. Although this does not treat the near-surface flows accurately, the main flow processes of interest (i.e. fountain flows or ground vortex/headwind interactions) were well away from the nozzle/intake geometry. Recent calculations using a multiblock, boundary conforming mesh [37] have confirmed the acceptability of this approximation.

Fig. 2 Schematic of the jet/intake assembly

3 DESCRIPTION OF THE FLOW PROBLEM

The numerical model has been applied to the experimental test data reported by Behrouzi and McGuirk [34] for the geometry shown in schematic form in Fig. 2 tested in an isothermal flow in a water tunnel to facilitate LDA measurements. The main flow parameter of interest is the jet/crossflow velocity ratio $R = V_j/W_c$. The corresponding velocity ratios (R) studied experimentally were 35, 24 and 18. The Reynolds number associated with these tests was around 40 000 ($V_j d_j/\nu$ or $W_c d_h/\nu$). The jet and crossflow velocities used in the water flow experiments were measured and found to be $V_j = 2.66$ m/s and $W_c = 0.076$ m/s ($R = 35$), 0.113 m/s ($R = 24$) and 0.15 m/s ($R = 18$) respectively. Although velocity ratios of 18 and 35 were studied, these yielded either insignificant ingestion events (18) or practically continuous ingestion (35). Hence, the $R = 24$ case was deemed to be the most suitable for the present study as corresponding to a case of highly intermittent ingestion, thus providing the most difficult test for steady state RANS predictions. Measurements were also reported with zero crossflow (no headwind and a velocity ratio of infinity).

LDA experiments were carried out in a specially designed and constructed water tunnel for STOVl flow applications. Figure 2 presents the geometry of the jet/intake unit, which was located centrally in the water tunnel and supported by the test section top wall. A single intake with a rounded lip (2:1 ellipse) was positioned between twin jets. The jet diameter, jet exit height, intake diameter and intake centre-line height were chosen as 12.5 mm (d_j), 87.5 mm ($7d_j$), 37.5 mm ($3d_j$) and 118.75 mm ($9.5d_j$) respectively. The jet spacing and intake position forward of the jets were both fixed at 75 mm ($6d_j$). The cross-section of the water tunnel was 0.3 m ($24d_j$) height by 0.37 m ($30d_j$) width. The presence of a (rudimentary) fuselage via a flat plate underneath the intake was allowed for in order to include possible forward deflection of the fountain. These dimensions and positions correspond approximately

to a geometrical configuration typical of near-ground aircraft operation. The intake suction pipe was connected directly to the jet pumps, which supplied the twin jet nozzles and were isolated from other circuits to ensure equality of the mass flowrate between the jets and intake.

Two test cases were selected for study: the first is referred to as an isolated twin jet system (i.e. no intake present) and the second corresponds to the combined twin jet plus intake configuration. Both of these geometries were predicted with and without the presence of a crossflow in the downstream water tunnel direction (W_c). The flowfield is symmetric about the intake centre plane (i.e. water tunnel mid-plane); therefore only half of the flowfield was modelled in the predictions. In the coordinate system used below to report the comparison between predictions and measurements (see Fig. 3), the origin of the longitudinal (z) coordinate (positive in the headwind or crossflow direction) is at the jet entry plane; the vertical (y) coordinate has its origin on the tunnel floor and is measured positive upwards; finally, the transverse (x) coordinate has its origin at the jet intake and water tunnel symmetry plane and is positive towards the front. The computational domain comprises a rectangular box with six boundaries: an inlet and outlet plane (left and right boundaries), a rear boundary symmetry plane and three solid walls at the top, bottom and front boundaries of the domain. The width and height of the solution domain were specified as $15d_j$ and $24d_j$ respectively (the same as the water tunnel test section dimensions). The effect of the location of the inlet and outlet planes on the solution had to be tested since this influenced the admissible use of simple boundary conditions in these planes (e.g. uniform flows or zero gradient conditions), but also affected the total number of grid points needed. After an optimization study, locations $30d_j$ upstream and downstream of the intake location were adopted. A non-uniform grid system of 276 210 nodes was employed ($n_x = 55$ in the transverse direction, $n_y = 54$ in the vertical direction and $n_z = 93$ in the streamwise direction). To examine whether this grid was fine enough

Fig. 3 Flowfield coordinate system

to provide solutions accurately, rather than to use a finer grid, numerical tests were conducted by comparing solutions with both first-order and second-order accurate discretization. If numerical errors on the mesh size chosen had been significant, then switching to a higher-order accuracy discretization would have produced a noticeable shift in the predicted solution. As shown in the next section, this was not observed; although some flow features were altered by the change in discretization accuracy, these changes were small enough to indicate that the solutions presented here are substantially grid independent. A uniform distribution of all dependent variables was prescribed at the inlet of the solution domain. Within the calculation domain the jet pipes and intake suction duct were replaced by simple internal blockages; wall functions were prescribed on both blockage and water tunnel walls. The intake flow was specified by fixing a uniform suction

velocity $3d_j$ inside the intake duct to ensure an intake flow equal to the jet nozzle discharge flow.

4 NUMERICAL RESULTS

Predictions of the flowfield were carried out for two velocity ratios of 24 and infinity. Results from first- and second-order numerical schemes are presented below to assess accuracy. All mean and r.m.s. turbulence velocity components are normalized by the jet exit mean velocity and all distances by the jet diameter (d_j).

Figure 4 depicts streakline pictures in (a) the fountain plane and (b) the jet discharge plane for test case 1 (i.e. the isolated twin-jet system) for a velocity ratio of infinity and using the second-order scheme. The flow is predicted to be

Fig. 4 Streakline pictures in (a) the fountain plane ($x/d_j = 0$) and (b) the jet discharge plane ($x/d_j = 3$) for an isolated twin-jet system with no crossflow

symmetrical about the fountain origin and jet impingement point, as expected. The fountain vertical penetration is about $14d_j$. Figure 5 presents the same results for test case 2 (i.e. the combined twin-jet intake system). The presence of the intake and its suction flow drastically changes the flow pattern in its immediate vicinity. The circulation region on the top of the fountain in Fig. 4a and in the jet plane in Fig. 4b (caused by the finite size of the water tunnel) has been removed by introduction of the intake. The entrainment flow into the jet has also been substantially modified. Finally, Fig. 5a indicates that direct ('short-circuit' type) reingestion occurs by deflection of the fountain flow forward into the intake (at infinity velocity ratio no ground vortex flow is found, of course).

Mean (W) and r.m.s. (w) profiles of axial velocity along the intake centre-line for two velocity ratios and both test case geometries are shown in Fig. 6. The mean and r.m.s. velocity distributions are flat for the case of zero intake

flow, but the presence of the intake suction induces significant flow accelerations within about two intake diameters in front of the intake. Figure 7 shows profiles of mean axial (W) and transverse (V) velocities and axial r.m.s. (w) along a vertical line in front of the intake ($x/d_j = 0$ and $z/d_j = -7$), again for both geometric configurations. The intake centre-line is located at $y = 9.5d_j$. The thickness of the ground sheet flow is predicted to be around $5d_j$ (Fig. 7a) and is almost unaffected by either introduction of the intake or the presence of crossflow. The effect of the intake on the flowfield becomes considerable above a height of $7d_j$. The crossflow creates a slightly wider capture area for the intake (Fig. 7a), which is identifiable by the presence of positive and negative vertical velocities at the lower and upper lip of the intake respectively (see Fig. 7b). The variations in axial turbulence intensity are associated with the interaction between the ground sheet flow and the

Fig. 5 Streakline pictures in (a) the fountain plane ($x/d_j = 0$) and (b) the jet discharge plane ($x/d_j = 3$) for a combined twin-jet/intake system with no crossflow

Fig. 6 Profiles of mean (W) and r.m.s. (w) axial velocity along the intake centre-line ($y/d_j = 9.5$ and $x/d_j = 0$) for both isolated twin-jet and combined twin-jet/intake systems

Fig. 7 Profiles of (a) mean axial and (b) mean transverse velocities and (c) axial r.m.s. along a vertical line in front of the intake ($x/d_j = 0$ and $z/d_j = -7$)

crossflow and changes in the strain rates associated with the acceleration induced by the intake suction. The former effect is again almost unaffected by the presence of crossflow, except of course at large heights above the tunnel floor; the latter effect is predicted to induce substantial turbulence generation in the intake vicinity (Fig. 7c), although the correctness of this is dependent on the way the

turbulence model reacts to flow accelerations (see comments below).

Figures 8 and 9 present predicted streakline pictures which illustrate the results of numerical accuracy tests on the test case 2 flow; vertical longitudinal planes through the intake (Fig. 8) and jet centre-lines (Fig. 9) are presented. The effect of switching from the first-order to

Fig. 8 Streaklines in the fountain plane ($x/d_j = 0$) at a velocity ratio of 24 for (a) the first-order scheme and (b) the second-order scheme

the second-order numerical scheme on the general flow pattern is not considerable. The major areas of difference are the predicted effect of crossflow on the fountain deflection and on the forward penetration (Figs 8a and b) and the size and shape of the ground vortex in front of the jets (Figs 9a and b). The fountain impingement point on the underside of the intake is deflected more downstream in the first-order prediction (Fig. 8a) than in the second-order results (Fig. 8b), where an almost vertical rise is predicted, as in the zero crossflow case. This implies that the direct route for ingestion will be stronger in the second-order results and hence shows the importance of numerical tests. The lower levels of numerical diffusion in the second-order results also enable the ground sheet to penetrate further forward (by around $2d_j$ —about 10 per cent) (see Figs 8a and b). The same is true for the ground vortex, which has shifted forward by about $2d_j$ in Fig. 9b compared to Fig. 9a

(a change of around 15 per cent). The size of the ground vortex has increased in both vertical and longitudinal directions in the second-order results, as has been observed before [15]. The gross features of the flow are otherwise very similar in the two calculations, so it is likely that the second-order predictions may be viewed as substantially grid independent.

A comparison of the present predictions with the LDA results of reference [34] is contained in Figs 10 to 13. Figure 10 shows results that capture the jet plume and fountain upwash characteristics at a distance $2d_j$ above the ground on the $z/d_j = 0.0$ plane. The benefits of the second-order accurate scheme can again be seen here with a better resolution of the peak jet velocities and improved predictions of the peak values in the turbulence field. The generation of turbulence in the separate jet shear layers is well resolved, as is the level of turbulence at the base of the

Fig. 9 Streaklines in the jet discharge plane ($x/d_j = 3$) at a velocity ratio of 24 for (a) the first-order scheme and (b) the second-order scheme

Fig. 10 CFD/LDA data comparison for the twin-jet flow plane ($y/d_j = 2$ and $z/d_j = 0$)

Fig. 11 CFD/LDA data comparison for decay of vertical velocity in fountain flow ($x/d_j = 0$ and $z/d_j = 0$)

Fig. 12 CFD/LDA data comparison in ground sheet flow ($y/d_j = 0.25$ and $x/d_j = 0$)

fountain. The entrainment between fountain flow and jet flow does not seem to be as well predicted, however, as indicated by the different shape of the transition region connecting these two features in Fig. 10. Figure 11 examines the accuracy of predictions of the decay of the fountain peak vertical velocity with height. Predictions with the numerically more accurate QUICK scheme show about the right decay rate but an overprediction of the velocities at all levels (note that the additional numerical diffusion provided by the first-order scheme produces an *apparently* better prediction, but this is misleading as far as assessment of the turbulence model performance is concerned). The results in Fig. 12 examine the quality of prediction of the ground sheet flow. It is clear that the forward penetration on this plane is greatly overpredicted by some $5d_j$, which is a serious error. The ability of an eddy-viscosity-based turbulence model to simulate decay of the forward flowing momentum in the ground sheet is evidently poor, stemming in all probability from errors produced at jet impingement, a well-known defect of eddy viscosity models, and also from large-scale unsteadiness at the furthest forward penetration point.

Figure 13 assesses the quality of the prediction of the intake flow itself by comparing with measurements of a

Fig. 13 CFD/LDA data comparison of intake profiles ($x/d_j = 0$ and $z/d_j = -7$)

vertical profile taken just in front of the intake ($z/d_j = -7$) on the symmetry plane $x/d_j = 0$. The peak vertical velocities induced by the intake are predicted to be far too high (Fig. 13a), and whereas the experiments show that the ground sheet flow and intake flow are separated by a region of low V velocity, these features have merged together in the predictions (too high a rate of momentum diffusion). The streamwise velocity profile also shows errors in both ground sheet and intake suction regions (Fig. 13b). There is

evidence here of the inability of an isotropic eddy viscosity model to capture the mean intake flow features adequately. For example, the calculated spread in the vertical (y) direction of the V velocity is too great, but is too small for the W velocity.

The response of the turbulence model to acceleration into the intake has produced large errors in the turbulence field (Fig. 13c). This is likely to be for the same reason that the standard linear $k-\epsilon$ eddy viscosity model produces too high a turbulence in the decelerating region near impingement. The presence of high strain rates always leads to high turbulence generation in the standard $k-\epsilon$ model, whereas if the Reynolds stresses are close to isotropic, this does not occur physically. Finally, flow visualization results of Behrouzi and McGuirk [34] have indicated that the jet impingement, forward ground vortex penetration and fountain regions are highly unsteady and are dominated by large-scale turbulent structures. Thus, attempts to model the turbulence in these regions via a Reynolds-averaged approach, which averages across all eddy sizes present, are likely to be inaccurate. One likely consequence is that in an RANS approach, the predicted rates of mixing of momentum in shear layers is less than the observed rate of spread, which is strongly influenced by large-scale flow oscillations (flapping). Hence there is an underprediction of fountain spread and too-large forward penetration. The present results indicate that accurate prediction of flows of this type is likely to require either time-averaged turbulence models, which are better able to describe the effects of impingement and streamline curvature on the turbulence isotropy (Reynolds stress models), or time-resolved (LES) simulations, which are needed to capture the large-scale unsteadiness in fountain and ground sheet/ground vortex regions.

5 CONCLUSIONS

As part of a programme of computational work on intake ingestion problems, the CFD prediction of the flowfield associated with a generic twin impinging jet plume/intake model was carried out and compared with available LDA experimental data. With the mesh density of a quarter of a million cells adopted, the use of second-order convection differencing was found sufficient to produce solutions close to grid independency. A reasonable prediction of the general flow pattern and the response to changing headwind conditions was achieved. The prediction of the velocity field in the majority of the flow region was in reasonable agreement with the LDA data. However, prediction of the turbulent fluctuations, especially near the intake and in the ground vortex forward penetration regions, was unsatisfactory. For steady state time-averaged predictions, the $k-\epsilon$ model was shown to produce errors in the prediction of the forward penetration distance of the ground flow and in the shape of velocity profiles and turbulence

levels near to the intake. Further work is required to eliminate possible errors arising from the use of a non-body-fitted mesh (although these are not believed to be of significance) and to examine whether moving to a Reynolds stress transport turbulence model or an LES-based approach will improve the quality of the predictions, particularly for the intake flow.

ACKNOWLEDGEMENTS

The research reported here has been supported by British Aerospace (Military Aircraft Division). The authors would like to thank British Aerospace for their financial support and Mr S. Rickman (BAe (MAD), Farnborough) in particular for his close interest in monitoring the development of the work.

REFERENCES

- 1 Kuhn, R. E. and Eshleman, J. Ground effects on V/STOL and STOL aircraft—a survey. AIAA paper 85-4033, 1985.
- 2 Stewart, V. R. and Kuhn, E. R. A method for estimating the propulsion induced aerodynamic characteristics of STOL aircraft in ground effect. NADC-80226-60, August 1989.
- 3 MacLean, R., Sullivan, J. and Murthy, N. B. Hot-gas environment around STOVL aircraft in ground proximity—Part 1: experimental study. *Am. Inst. Aeronaut. Astronaut. J. Aircraft*, January–February 1992, **29**(1), 67–72.
- 4 Hall, G. R. and Rogers, K. H. Recirculation effects produced by a pair of heated jets impinging on a ground plane. NASA CR-1307, May 1969.
- 5 Bower, W., Saripalli, K. and Agarwal, R. A summary of jet impingement studies at McDonnell Douglas Research Labs. AIAA paper 81-2613, 1981.
- 6 Stewart, V. R. and Kuhn, R. E. Characteristics of the ground vortex developed by various V/STOL jets at forward speed. AIAA paper 83-2494, October 1983.
- 7 Kuhn, R. E. The induced aerodynamics of jet and fan powered V/STOL aircraft. In *Recent Advances in Aerodynamics*, 1986, pp. 337–373 (Springer-Verlag, New York).
- 8 Kuhn, R. E. Hot-gas ingestion and the speed needed to avoid ingestion for transport type STOVL and STOL configurations. AIAA paper 84-2530, 1984.
- 9 Agarwal, R. K. Recent advances in prediction methods for jet-induced effects on V/STOL aircraft. In *Recent Advances in Aerodynamics*, 1983, pp. 471–521 (Springer-Verlag, New York).
- 10 Kotansky, D. R. Multiple jet impingement flow fields. Special Course on V/STOL Aerodynamics. AGARD Report 710, 1984, pp. 7-1–7-48.
- 11 Bradbury, L. J. S. Some aspects of jet dynamics and their implications for VTOL research. AGARD Report CP-208, 1981.
- 12 Hancock, G. J. A review of the aerodynamics of a jet in a cross-flow. *Aeronaut. J.*, 1987, **21**(905), 201–213.
- 13 Barata, J. M. M., Durao, D. F. G., Heitor, M. V., et al. Experimental and numerical study of the aerodynamics of jets

- in ground effect. In Tenth Symposium on *Turbulence*, University of Rolla-Missouri, 1986.
- 14 Behrouzi, P. and McGuirk, J. J. CFD investigation of single subsonic jet in unconfined crossflow for ASTOVL flow application. *Iranian J. Sci. Technol.*, 1993, **17**(3), 179–189.
 - 15 Barata, J. M., Durao, D. F. G. and McGuirk, J. J. Numerical study of single impinging jets through a crossflow. *Am. Inst. Aeronaut. Astronaut. J. Aircraft*, 1989, **26**(11), 1002–1008.
 - 16 Jones, W. P. and McGuirk, J. J. Computation of a round turbulent jet discharging into a confined cross-flow. In *Turbulent Shear Flows*, Vol. 2, 1979, pp. 233–245 (Springer-Verlag, New York).
 - 17 Chien, C. J. and Schetz, J. A. Numerical solution of the three-dimensional Navier–Stokes equations with application to channel flows and buoyant jet in a cross-flow. *Trans. ASME, J. Appl. Mechanics*, 1975, **42**, 575–579.
 - 18 Patankar, S. V., Basu, D. K. and Alpay, S. A. Prediction of the three-dimensional velocity field of a deflected turbulent jet. *Trans. ASME, J. Fluids Engng*, 1977, **99**(4), 758–762.
 - 19 Baker, A. J., Orzechowski, J. A., Manhardt, P. D. and Yen, K. T. A three-dimensional finite element algorithm for prediction of V/STOL jet-induced flowfields. AGARD Report CP-308, 1982.
 - 20 Sykes, R. I., Lewellen, W. S. and Parker, S. F. On the vorticity dynamics of a turbulent jet in a crossflow. *J. Fluid Mechanics*, 1986, **168**, 393–413.
 - 21 Baker, A. J., Snyder, P. K. and Orzechowski, J. A. Three dimensional nearfield characterisation of a VSTOL jet in turbulent crossflow. AIAA paper 87-0051, 1987.
 - 22 Roth, K. R. Numerical simulation of a subsonic jet in a crossflow. SAE paper 872343, 1987.
 - 23 Harloff, G. J. and Lytle, J. K. Three-dimensional viscous flow computations of a circular jet in a subsonic and supersonic cross flow. AIAA paper 88-3703, 1988.
 - 24 Khan, Z. A., McGuirk, J. J. and Whitelaw, J. H. A row of jets in a cross flow. AGARD Report CP-308, paper 10, 1981.
 - 25 Demuren, A. O. Numerical calculations of steady three-dimensional turbulent jets in cross flow. *Comput. Meth. Appl. Mech. and Engng*, 1983, **37**, 309–328.
 - 26 Claus, R. W. Numerical calculation of subsonic jets in crossflow with reduced numerical diffusion. AIAA paper 85-1441, 1985.
 - 27 Childs, R. E. and Nixon, D. Simulation of impinging turbulent jets. In AIAA 23rd Aerospace Science Meeting, Reno, Nevada, 14–17 January 1985.
 - 28 Marquis, A. J. The application of high order convection approximations and second order turbulence closures to recirculating flows. PhD thesis, University of London, 1986.
 - 29 Claus, R. W. and Vanka, S. P. Multigrid calculations of a jet in cross flow. In AIAA 24th Aerospace Science Meeting, Reno, Nevada, January 1990, paper 90-0044.
 - 30 Van Overbeke, T. J. and Holdeman, J. D. A numerical study of the hot-gas environment around a STOVL aircraft in ground proximity. AIAA paper 88-2882, 1988.
 - 31 Van Overbeke, T. J. and Holdeman, J. D. Three-dimensional turbulent flow code calculations of hot-gas ingestion. *Am. Inst. Aeronaut. Astronaut. J. Aircraft*, 1990, **27**(7), 577–582.
 - 32 Fricker, D. M., Holdeman, J. D. and Vanka, S. P. Calculation of hot-gas ingestion for a STOVL aircraft model. *Am. Inst. Aeronaut. Astronaut. J. Aircraft*, 1994, **31**(1), 236–242.
 - 33 Tafti, D. K. and Vanka, S. P. Hot gas environment around STOVL aircraft in ground proximity—Part 2: numerical study. *Am. Inst. Aeronaut. Astronaut. J. Aircraft*, 1993, **29**(1), 20–27.
 - 34 Behrouzi, P. and McGuirk, J. J. Experimental data for CFD validation of the intake ingestion process in STOVL aircraft. Paper accepted for publication in *Flow, Turbulence and Combustion*, 1999 (Kluwer Academic Publishers, Dordrecht, The Netherlands).
 - 35 Launder, B. E. and Spalding, D. B. *Mathematical Models of Turbulence*, 1972 (Academic Press, London).
 - 36 Leonard, B. P. A stable and accurate convective modelling procedure based on quadratic upstream interpolation. *Comput. Mech. Engng*, 1979, **19**, 59.
 - 37 Page, G. J., McGuirk, J. J. and Jiang, D. Application of computational fluid dynamics to hot gas ingestion modelling. In RAe Soc International Powered Lift Conference, London, September 1998.