NUMERICAL INVESTIGATION OF PARAMETERS AFFECTING THE THERMAL AND HYDRODYNAMIC CHARACTERISTICS OF IMPINGING JETS IN CROSS FLOW

B. M. Suloiman

Department of Mechanical Engineering, University of Jordan, Amman-Jordan

B. A. Jubran

Department of Mechanical and Industrial Engineering, Sultan Qaboos University, Al-Khod-Sultanate of Oman

***Abstract*:** In this investigation the hydrodynamic and the thermal fields due to a single impinging jet in cross-flow have been investigated numerically, using a 2-D axisymmetric model in order to predict the ground vortex characteristics. The parameters investigated include the effective velocity ratio, the nozzle height, the nozzle pressure ratio, the intake location, the intake mass flow rate and the jet temperature ratio. It is interesting to note that even with the 2-D modeling limitations it was possible to capture most of the thermal and fluid field characteristics of the ground vortex. It was found that the temperature distribution in the flow field is greatly affected by the effective velocity, and the maximum penetration point of the ground vortex is equal to the hot gas penetration. The ground vortex strength increases slightly with increasing the intake mass flow rate but has a minor effect on the ground vortex geometry and on the penetration of the hot gases. The intake location has a significant effect on the ground vortex strength when it is located upstream of the ground vortex core.

Key Words: Numerical investigation, Turbulence Models, Impinging jets, Cross-flow.

1. INTRODUCTION

Jets in cross flow are encountered in many engineering applications, such as cooling of gas turbine blades and combustion chambers, plume dispersion, reaction control for missiles, flow of waste water into rivers, and vertical and/or short takeoff/landing (V/STOL) aircrafts. An extensive bibliography of numerous experimental and numerical investigations on jets in cross flow has been presented by Margason [1].

Modeling of impinging jets in cross flow with particular application to V/STOL aircraft seems to be centered around the hydrodynamic characteristics of the flow with a limited number of investigations on the thermal characteristics associated with such a flow. One could classify these investigations into basically three groups; the first group deals with the validation of various turbulence models, the second group tests the potential of various solution procedures and the third group investigates the effects of various parameters on the flow field of such a problem. Jones and McGuirk [2], and Childs and Nixon [3] modeled turbulent round jets in cross flow using three-dimensional grids and the standard k-ε model with little success. Van Dalsem et al [4] investigated the ground vortex associated with a single jet in cross flow using the Baldwin-Lomax algebraic turbulence model and the predicted results agreed reasonably well with the experimental results.

Catalano et al [5] modeled a turbulent jet in a confined cross-flow using a power law scheme to account for the combined effects of convection and diffusion between the adjacent grid points. A staggered grid was used with the pressure and the other dependent variables stored in the main grid points while the velocities were stored in the staggered locations. The predicted and the experimental results show good agreement except in the vicinity of the jet where the prediction was relatively poor due to the anisotropic nature of the flow in this region. Barata et al [6] used the QUICK scheme with a standard k-ε model to predict the flow field of jet in cross flow and found that such a model was not able to correctly predict the shear stress distribution in the impinging zone.

Hwang and Liu [7] solved the Reynolds-averaged compressible Navior-Stokes equations together with the standard k-ε turbulence model to predict the impinging jet flow fields associated with flow fields associated with V/STOL aircrafts. Implicit finite difference schemes were used with implicit boundary treatment to predict the effects of the height of the nozzle on the flow field of a jet in cross flow. Agarwal and Bower [8] solved the Navier Stokes and energy equations in a stream function/vorticity form in conjunction with the k-ε model together with an augmented central difference scheme to preserve the diagonal dominance character of the difference equations at high Reynolds numbers. The resulting difference equations were solved by successive point relaxation. The main conclusion of this investigation is that k-ε model has shortcomings in predicting the thermal as well as the hydrodynamic field of impinging jets relevant to V/STOL aircrafts.

Hwang et al [9] investigated numerically the airfoil-jet-ground interaction flow field and planar jet issuing from fuselage under surface in ground effect. Implicit-factored scheme with central difference and explicit

boundary were used to solve the averaged compressible Navier-Stokes equations for two-dimensional flow that is related to V/STOL aircraft. The cross-flow Reynolds numbers used were 5,000 and 20,000 while temperature ratio between the jet and the cross flow were 1 and 1.5. It was found that when the temperature of the jet was increased, the jet strength was reduced due to the decrease in the density at the plane of the entering jet.

Bray and Knowles [10] conducted a parametric study for some of the parameters affecting the flow field of impinging jet in cross flow using a standard k-ε model. Their predicted results indicate that the vortex core and the maximum penetration points are coincident with the ground plane minimum and maximum penetration positions. One of their main conclusions is that the effects attributed to the jet height are due to inaccuracies in the modeling of the free jet turbulence. Smith et al [11] investigated the jet flow fields that could be generated by V/STOL aircraft in ground effect using two commercial packages called FLOSYS and PHOENICS. Both packages predicted the experimental result rather well and found the *k-ε* model turbulence model tends to increase the mixing of the jet. The nozzles temperature effects were also investigated and it was found that the predicted temperature in the intake was increased by approximately 180°C at h/d=4.

The hot gas environment around a V/STOL aircraft operating in ground proximity was numerically modeled by Van Overbeke and James [12] and Tafti and Vanka [13] using the same configuration of multiple impinging jets with temperature fields close to the engine inlet. The predicted results indicate that the mean inlet temperature rise increased with decreasing the head wind and decreasing distance from the ground. Furthermore, the hot gas ingestion (HGI) from the fountain flow was more severe than the HGI due to the recirculating flow.

It appears from the aforementioned investigations that most attention has been paid to the flow field characteristics rather than the thermal field. It is hoped that this paper will provide an extended parametric investigation, which will include both hydrodynamic as well as thermal parameters of the flow with some emphasis on the parameters that affect the HGI.

2. MATHEMATICAL FORMULATION

A schematic diagram showing the flow field that would result from the impingement of a jet normal to the surface in the presence of a cross flow is shown in figure 1. The general governing equation for a single-phase, two-dimensional, steady state flow is given by [14]:

This equation may be written for plane or axi-symmetric geometry as

$$div(\rho \vec{V}\phi) = div(\Gamma grad\phi) + S_\phi \tag{1}$$

and

$$\frac{\partial}{\partial x}\left[\rho r^j u\phi\right] + \frac{\partial}{\partial y}\left[\rho r^j v\phi\right] = \frac{\partial}{\partial x}\left[r^j \Gamma^\phi \frac{\partial}{\partial x}\right] + \frac{\partial}{\partial y}\left[r^j \Gamma^\phi \frac{\partial}{\partial y}\right] - r^j S_\phi \tag{2}$$

where the co-ordinates are defined in terms of *x* and *y* directions with *y=r* for the axi-symmetric geometry; *u* and *v* are the local velocities in the *x* and *y* directions, respectively. As was pointed out by Bray and Knowles [10] that although the problem is inherently unsteady 3-D flow field, yet 2-D modelling is capable of providing an economical parametric investigation. Three-dimensional modeling can then be used to further investigate some selected conditions. The turbulent exchange coefficient and source terms for the general variable ϕ are Γ_ϕ and S_ϕ, respectively. The superscript *j=1* for the axi-symmetric case and *j=0* for the plane case. The expression for the turbulent exchange coefficient and source terms are given for the various solved equations in table 1. The compressibility effect was introduced into the code by using the perfect gas equation (ρ=p/RT), where p, T, and R are the pressure, temperature, and the gas constant of the fluid, respectively.

Table 1: Summary of the coefficients and sources of the variable ϕ

Equations	ϕ	Γ^ϕ	S_ϕ
Continuity	1	0	0
-momentum	u	μ_{eff}	$-\frac{\partial \rho}{\partial x} + \frac{\partial}{\partial x}\left(\mu_{eff}\frac{\partial u}{\partial x}\right) + \frac{1}{r^j}\frac{\partial}{\partial y}\left(r^j \mu_{eff}\frac{\partial v}{\partial x}\right)$
			$-\frac{\partial \rho}{\partial y} + \frac{\partial}{\partial x}\left(\mu_{eff}\frac{\partial u}{\partial y}\right) + \frac{1}{r^j}\frac{\partial}{\partial y}\left(r^j \mu_{eff}\frac{\partial v}{\partial y}\right) - \frac{2j\mu_{eff} v}{r^2}$
y-Momentum	v	μ_{eff}	
Turbulence Energy	k	μ_{eff}/σ_k	$G - \rho\varepsilon$
Dissipation	ε	$\mu_{eff}/\sigma_\varepsilon$	$\frac{\varepsilon}{k}(C_1 G - C_2 \rho\varepsilon)$
Energy	T	μ_{eff}/σ_T	0

The standard k-ε model is used in the present investigation where Reynolds stress is computed using the Boussineseq eddy viscosity concept, which is given by

$$-\rho\overline{u_i u_j} = \rho\nu_t(u_{i,j} - u_{j,i}) - \frac{2}{3}\rho k\delta_{ij} \qquad (3)$$

The eddy viscosity ν_t is found from $\nu_t = C_\mu C_D k^2/\varepsilon$ where k, and ε are the turbulent kinetic energy and its rate of dissipation, respectively. The two transport equations used for the solutions of k *and* ε are given by

$$(\rho k)_{,t} + [\rho u_i k - \rho\frac{\nu_t}{\sigma_k}k_i]_{,i} = \rho(P_k - \varepsilon) \qquad (4)$$

$$(\rho\varepsilon)_{,t} + [\rho u_i \varepsilon - \rho\frac{\nu_t}{\sigma_\varepsilon}\varepsilon_i]_{,i} = \rho\frac{\varepsilon}{k}(C_{1\varepsilon}P_k - C_{2\varepsilon}) \qquad (5)$$

where P_k is the volumetric production rate of the kinetic energy and is given by $P_k = -u_i u_j U_{i,j}$, and the standard values of the unmodified empirical constant are; $C_\mu C_D$=0.09, $C_{1\varepsilon}$=1.44, and $C_{2\varepsilon}$=1.92, σ_k=1.0, and σ_ε=1.314.

Adjacent to the wall, the model of turbulence must account for the viscous effects and, to do that, the wall function is used. The wall function may be written as follows:

$$U^+ = \frac{1}{K}ln(Ey^+) \qquad (6)$$

$$k_w = U_\tau^2(C_\mu C_D)^{-1/2} \qquad (7)$$

$$\varepsilon_w = \frac{U_\tau^3}{Ky} \qquad (8)$$

where $U^+ = U/U_\tau$, $U_\tau = (\tau_w/\rho)^{0.5}$, $y^+ = U_\tau^y/\nu_\ell$, K=0.435, E=0.9 and ν_ℓ is the laminar kinematic viscosity.

The flow equations were solved using the embodied SIMPLE procedure. Additional information on the mathematical procedures used may be found into the TEAM manual [14].

Fig. 1 Flow field of turbulent impinging jet in cross-flow

3. BOUNDARY CONDITIONS AND COMPUTATIONAL DETAILS

The elliptic nature of the problem in question required the specifications of the boundary conditions along all domain boundaries for all dependent variables. The boundary conditions used in this problem may be divided into five types; fluid entry plane where all flow properties must be known and prescribed (ϕ = constant), axis of symmetry where the gradients of all properties normal to this axis are zero, fluid exit plane where zero streamwise gradient of all properties are used with a uniform pressure along the exit plane, walls where the velocity components and the turbulent quantities are zero with the use of the wall function, and the entrainment boundary where the pressure is uniform and the tangential velocity component is set to zero. Preliminary numerical tests were conducted to explore the grid dependence and the number of iterations. It was found that an independent solution is obtained with a grid of 52x34 for the case without the intake effects and 58x34 for the case with the intake. The grids were generated in such a way that very close to the walls the density of the grid points is high. The grid used in modelling is shown in figure 2.

Fig. 2a Boundary conditions for modeling impinging jet in cross-flow

Fig. 2b Typical grid used for modeling impinging jet in cross-flow

A convergence criteria of the residual of 1% was typical for most of the studied cases. However, the two most difficult variables to converge were the kinetic energy k and the dissipation rate ε and for these two variables to converge within the 1%, the residuals for the other quantities were usually below around 0.01%. The present predictions were carried out on a VAX 8700 computer at the University of Jordan. A typical number of iterations for a case without intake is around 5000 iterations, while for the intake case is around 9000 iterations.

4. RESULTS AND DISCUSSION

The predicted results will be compared with the experimental and the numerical results of Bray [15]. The main parameters which have been investigated include the effective velocity V_e the nozzle height to diameter ratio h/d, the pressure ratio Pr, *the* jet temperature to freestream temperature, T_j/T_∞ ratio, the intake mass flow rate to the jet mass flow rate m_i/m_j and the location of the intake ℓ/d on the flow field of interest. A summary of the values of parameters investigated is shown in table 2.

Table 2. Summary of parameters used in the numerical modeling

Parameter	Values
V_e	14.4, 17.9, 25.1 and 26.1
Pr	1.05, 1.2, 1.5, and 1.6
h/d	2.0, 2.5, 4.0, 5.0, 6.0,7.5, 8.0 and 10
T_j/T_∞	0.89, 2.4 and 3.2
m_i/m_j	0 (no intake), 0.25, 0.5, 1.0, 1.5, 2.0, 2.5, 3.0, and 4.0
ℓ/d	10.5, 13.5, 16.5 and 22.5

Figure 3 shows the velocity vector plots when h/d=5.0, Pr=1.5, T_j/T_∞=3.2, and V_e=25.1. The pressure coefficient distributions at different effective velocities are shown in figure 4(a). It can be seen from these figures that the minimum C_p position at x/d=13.5 coincides with the vortex core position while the maximum C_p position coincides with the maximum penetration position. These figures also show that when the jet impinges on the ground plane, it stagnates and deflects forward, forming a wall jet that meets the freestream where a stagnation point is formed and the wall jet is deflected backwards. Part of this back-flowing fluid is entrained again into the wall jet forming a ground vortex region, figure 3.

The predicted results of the horizontal distances of the vortex penetration x_p and the ground vortex core x_v reveal the existence of a fixed relation which can be determined from the static pressure distributions for all the cases investigated. The predicted relation is given by x_p/x_v=1.58 compared with the experimental relation of Bray [15], which is x_p/x_v=1.592. It is interesting to note that although both of x_p and x_v are overpredicted when compared with the experimental ones, the deviation of the experimental and predicted relations is about 0.75%. This numerical correlation is independent of the flow parameters and indicates that the ground vortex can be described by its characteristic locations. Another correlation which is useful is the relation between the vertical and horizontal locations of the penetration of the vortex and is found to be given by y_v/x_v=0.184.

One of the important parameters that affect the ground vortex geometry is the effective velocity ratio V_e. This is due to the fact that the maximum penetration distance increases with increasing the effective velocity as can be seen from figure 4(a) where it can be seen that the position of C_{pmax} is increased with increasing V_e. The rate of increase of x_p/d *with* V_e is nearly independent of the nozzle height as it is indicated in figure 4(b). For all cases studied the predicted correlation is given by $x_p/d=0.86V_e$. The ground vortex size, which is indicated by, y_v/d shows similar trends to x_v and the relation between the vortex core height and V_e is given by $y_v/d=0.1V_e$. This correlation is independent of the nozzle height for the range investigated, h/d=2.0-10.0. The trend of increasing x_p/d *with* V_e was found in all previous numerical and experimental studies, such as [15-17]. Nevertheless, there is a significant disagreement among investigators regarding the rate of increase of x_p/d. This could be partially attributed to the different freestream boundary layer thickness used in the various investigations. A comparison between the present predictions with the experimental and numerical results of Bray [15] is shown in figure 5. The rate of increase of x_p/d *with* V_e is well predicted, but the absolute values of x_p/d are overpredicted. The present predictions are much closer to the experimental results. The present prediction overpredicts the vortex penetration by about 25% whilst Bray's prediction overpredicts the vortex penetration by about 40%. This deviation between the two numerical predictions may be attributed to the different numerical schemes used to interpolate the values of the scalar variables at the control volume faces. While Bray used the PLDS scheme for all variables, the present authors used the QUICK scheme for the velocity components and the PLDS scheme for all other scalar variables. It is well known that the QUICK scheme tends to cause less numerical diffusion than the PLDS scheme. However, the difference between the experimental and the predicted vortex penetrations may be due to the deficiency of the $k-\varepsilon$ turbulence model as well as the two-dimensional modeling of the problem.

The effect of height h/d on the vortex penetration is seen to be insignificant from the predicted results of the pressure distributions, figure 6. A comparison between the present predicted results and Bray's experimental and predicted results is shown in figure 7 which shows that the predicted results are not consistent with the experimental results which indicate that there is a slight increase in the vortex penetration with the increase in the height until a critical height is reached, after which there is a slight decrease in vortex penetration. One argument reported by Bray[15], which attributed the effects of height, obtained experimentally to the rig interference. Similar trends to the present work regarding the effects of height on the vortex penetration were reported by Tafti and Vanka[13].

Fig. 3 Velocity vector plot of the ground vortex region

Fig. 4a Effect of V_e on C_p distribution at $h/d = 5$

Fig. 4b Effect of V_e on x_p/d at different values of at h/d

Fig. 5 Comparison between the experimental and predicted results of x_p/d at $h/d = 5$

Fig. 6 Effect of h/d on C_p at distribution of at $V_e = 25.1$

Fig. 7 Comparison between the experimental and predicted results of x_p/d at various h/d and V_e values

It seems that very little work has been done regarding the effects of the jet temperature on the flow field of jets in cross-flow relevant to V/STOL aircraft. In order to shed some light on this, a number of runs at different temperature ratios were obtained. Figures 8(a-b) show the effects of the temperature ratio on both the pressure coefficient distributions as well as on the vortex penetration. It is very clear that the effect of the jet temperature is rather minor at all effective velocity ratios. It is worth noting that increasing the jet temperature decreases the jet density and consequently decreases the jet dynamic pressure leading to a decrease in *Ve*. If the freestream velocity was decreased in such a way that V_e stays constant, then there will be no effect of the temperature since the temperature effect is included in V_e. This makes V_e the proper non-dimensional parameter that must be used to model the flow field since it combines both, the effect of the jet to cross-flow velocity ratio together with the effect of compressibility that arises from using different jet and/or cross flow temperatures. This argument was first observed experimentally by Abbot [17] who investigated the effect of different jet temperatures and found that the vortex penetration is independent of temperature for the same V_e. These findings were also supported by Corsiglia et al. [18] who found that the effect of jet temperature on the pressure data was small.

The effect of engine inlet conditions on the vortex location and geometry also did not receive enough attention in previous numerical investigations. The effect of inlet was mainly studied experimentally. MacLean et al.[19] conducted an experimental investigation on a typical model configuration. The test rig consisted of two inlet configurations with four jets impinging on a flat plate. It was found that the inlet suction appeared to increase the ground vortex location only at low height and low freestream to jet velocity ratios, V_∞/V_j In order to study the effect of the engine inlet conditions numerically on the ground vortex, one has to study the two parameters that are related to the inlet conditions, namely; the ratio of the engine inlet mass flow rate to the jet mass flow rate m_i/m_j, and the horizontal distance between the engine inlet and the nozzle jet location ℓ / d .

The two-dimensional modeling used in the present investigation necessitated that the inlet condition was modeled as a horizontal mass sink on the top of the free surface at a height of 10 d above the ground plane. The mass flow rate of the engine inlet was prescribed in such away to satisfy the ratio of mi/m_j.

Brady and Ludwig [20] found that the ground board static pressure distribution is independent of mass flow rate. The predicted results shown in figures 9(a-b) support such findings where it can be seen that the effect of m_i/m_j on the C_p distribution in terms of the positions of Cpmin and C_{pmax} is negligible. This small effect can be seen from the increase in C_{pmin} *as* m_i/m_j is increased which consequently increases the vortex strength except for the case of m_i/m_j =0 (no intake). However, the size of the ground vortex does not change with increasing m_i/m_j as long as V_e and ℓ/d are the same. Figure 10 shows that x_p/d decreases only slightly with increasing m_i/m_j.

In the present investigation, the intake location ℓ/d was varied from 10.5 to 22.5 for a fixed value of m_i/m_j =1. Figure 11(a) shows that when ℓ/d = 10.5 and 13.5, the C_p plots are nearly the same in terms of the positions and the values of C_{pmin} and C_{pmax} which means that the ground vortex location and strength are the same for these two cases. Figure 11(b) indicates that the location of the inlet of the engine has a minor effect on x_p/d where it is increased rather slightly as ℓ / d is increased. When higher values of ℓ/d such as 16.5 and 22.5 are used as can be seen from figure 12(a), the C_p plots are different in terms of C_{pmin} and C_{pmax}. For both cases, C_{pmin} is low compared with the cases of ℓ/d = 10.5 and 13.5 which means that the ground vortex is weak. When ℓ/d=22.5, C_{pmax} is lower than that of the case ℓ/d =16.5. The velocity vectors shown in figure 12(b) reveal that for ℓ/d=22.5, there is a clear ground vortex and the self-similarity laws are applicable. However, the ground vortex is weak and the stagnation pressure is low, as indicated from the C_p plots, figure 12(a). This may be explained by the fact that for ℓ/d=22.5, the cross flow is strongly deflected upwards towards the upper free surface due to the existence of the suction of the inlet, figure 12(b). This strong deflection leads to a reduction in the horizontal component of the cross-flow momentum and consequently leads to a reduction in collision losses with the wall jet and hence the stagnation pressure will be reduced and the ground vortex strength will be also reduced.

From the above argument one could conclude that when the intake is positioned ahead of the ground vortex core, the suction effect of the intake will tend to modify the flow structure and hence leads to weaker vortices and/or lower C_{pmax}. However, it must be noted here that it is possible that the 2-D modeling contributed to this flow behavior, since the flow cannot be deflected sideways and it is forced to deflect upwards.

The next set of predicted results is the normalized temperature $T_\theta=(T-T_\infty)/(T_j- T_\infty)$ and is presented as temperature contours in order to investigate the temperature distribution of the hot gases in the flow field. Figure 13(a) shows the temperature contours when T_j/T_∞= 3.2, V_e=14.4, m_i/m_j=1, and h/d =5. It can be clearly seen that at the centerline of the free jet region, the temperature decreases steadily with the vertical distance from the nozzle. At the jet exit, T_θ =1.0, while at the ground plane T_θ = 0.6, which means that the jet temperature is reduced by 40%. Similarly, for the wall jet, T_θ decreases steadily with the distance from the impingement point where it eventually reaches a zero value. The effective velocity ratio V_e has a significant effect on the temperature contours, where the penetration of the hot gases increases with increasing V_e. It can be seen from figure 13(b) when V_e =17.9 that the hot gases in the wall jet penetrates to a greater distance than those shown in figure 13(a) for V_e =14.4.

Fig. 8a Effect of T_j/T_∞ on C_p distribution

Fig. 8b Effect of T_j/T_∞ on x_p/d at different values of V_e

Fig. 9a Effect of low values of m_i/m_j on C_p distribution

Fig. 9b Effect of high values of m_i/m_j on C_p distribution

Fig. 10 Effect of m_i/m_j on x_p/d

Fig. 11a Effect of low values of l/d on C_p distribution

Fig. 11b Effect of l/d on x_p/d distribution

Fig. 12a Effect of high values of l/d on C_p distribution

Fig. 12b Velocity vector plot at $l/d = 22.5$

Fig. 13a Temperature contours at $T_j/T_\infty = 3.2$ and $V_e = 14.4$

Fig. 13b Temperature contours at $T_j/T_\infty = 3.2$ and $V_e = 17.9$

5. CONCLUSION

Several points have emerged from the numerical modeling of impinging jet in cross-flow with particular reference to V/STOL aircraft. These can be summarized as follows:

1. The effective velocity ratio is the most predominant parameter that affects the ground vortex geometry and the ground vortex strength increases with V_e.
2. The ground vortex similarity relation is predicted with high accuracy. The predicted relation is x_p/x_v=1.58 compared with 1.59 as obtained from experiment.
3. The nozzle height has little effect on the ground vortex, location and strength but it has a significant effect on the temperature at the impinging point where the temperature significantly decreases with increasing the height due to the entrainment action. However, the penetration of the hot gases does not change significantly with increasing the height.
4. The jet temperature has a negligible effect on the ground vortex at constant V_e and this effect is included in the effective velocity. Furthermore, the jet temperature has a negligible effect on the temperature distribution of the flow field.
5. The intake mass flow rate has a small effect on the ground vortex geometry and location. However, the ground vortex strength increases as the intake mass flow rate increases.
6. The intake location has a negligible effect on the ground vortex if it is positioned above or downstream of the ground vortex core. If it is positioned upstream of the ground vortex core, the strength of the ground vortex decreases. If the intake is positioned upstream of the maximum penetration point, the stagnation pressure is highly reduced. However, the HGI is increased as ℓ/d is decreased. The location of the intake has a negligible effect on the temperature distribution in the flow field.

REFERENCES

[1] R. J. Margason, "Fifty years of jet in cross flow research, "AGARD conference proceedings, Computational and Experimental Assessment of Jets in Cross Flow, Symposium held in Winchester, United Kingdom, 19-22 April, 1993.

[2] W. P. Jones, and J. J. McGuirk, "Computation of a round turbulent jet discharging into a confined cross-flow," Turbulent shear flows 2, Springer-Verlag, Berlin, pp. 233-245, 1980.

[3] R. E. Childs, D. Nixon, "Turbulent and fluid/acoustic interaction in impinging jets," Int. Powered Lift Conf., P-203, pp. 447-458, paper No. 872345, Santa Clara, Calif., Feb. 1988.

[4] W. R. Van Dalsem, A. G. Panaras, and J. L. Steger, " Numerical investigation of a jet in ground effect with a cross-flow," Int. Powered Lift Conf., P-203, pp. 447-458, paper N0. 872344, Santa Clara, Calif., Feb. 1988.

[5] G. D. Catalano, K. Chang, and J. A. Mathis, "Calculation and measurement of turbulent jet impingement in a cross-flow, Part 1., "6th International Conference on Numerical Methods in Laminar and Turbulent Flow, Vol. 6, pt.2, pp.1895-1906, Pineridge press, 1989.

[6] J. M. Barata, D. F. Durao, and J. J. McGurik, Numerical study of a single impinging jets through a cross-flow", Int. J. Aircraft, Vol. 26, No. 11, 1989.

[7] C. J. Hwang, and J. L. Liu, "Numerical study of two dimensional impinging jet flow field," AIAA Journal, Vol. 27, No. 7, 1989.

[8] R. K. Agarwal and W. W. Bower, "Navier Stokes computations of compressible two dimensional impinging jet flow fields," AIAA Journal, Vol. 20, No. 5, pp. 577-584, 1982.

[9] C. Hwang, S. Yang and J. Liu, "Numerical investigation of airfoil-jet fuselage undersurface flow field in ground effect," 28th Aerospace Science Meeting, AIAA 90-0597, Reno, Nevada, January 9-11, 1990

[10] D. Bray, and K. Knowles, "Numerical modeling of impinging jet in cross-flow," AIAA/ASME/ASEE 26th Joint Propulsion Conference, AIAA, 90-2246, Ornaldo, July 16-18, 1990.

[11] A. G. Smith, D. N. Ing, and P. J. Baily, "The experimental and computational study of jet impingement flow fields with reference to VSTOL aircraft to VSTOL aircraft performance," International Powered Lift Conference, 29-31 August, 1990.

[12] T. J. Vanoverbeke, and D. James, " A numerical study of the hot gas environment around a STOVL aircrafts in ground proximity," AIAA/ASME/SAE/ASEE 24th Joint Propulsion Conference, AIAA-88-2882, Boston, Massachusetts, July 11-13, 1988.

[13] D. Tafti and S. P. Vanka, "Hot gas environment around STOVL aircraft in ground proximity, part 2: Numerical study," ," AIAA/ASME/SAE/ASEE 26th Joint Propulsion Conference, AIAA-90-2270, Orlando, FI., and July 16-18, 1990.

[14] P. G. Huang and M. A. Leschziner, "An introduction and guide to the computer code TEAM," University of Manchester, 1992.

[15] D. Bray, "Jets in cross-flow and ground effect", Ph.D. thesis, Cranfield Institute of Technology, RMCS, England, 1991.

[16] J. M. Cimbala, D. R. Stingebring, A. L. Treaster, M. L. Billet and M. M. Walters," Experimental investigation of a jet impinging on a ground plane in the presence of a cross-flow," International Powered Lift Conference, December 7-10, 1987.

[17] W. A. Abbott, Studies of flow fields created by vertical and inclined jets when stationary or moving over a horizontal surface," Aero. Research Council, Rept. No. CP-911, October 1964.

[18] V. R. Corsiglia, D. A. Wardwell and R. E. Kuhn, "Small-scale experiments in STOVL ground effects," International Powered Lift Conference, 29-31 August 1990.

[19] R. Maclean, J. Sullivan and S. N. Murthy, "Hot gas environment around STOVL aircraft in ground proximity, part 1: Experimental study," AIAA/ASME/ASEE 26th Joint Propulsion Conference, AIAA 90-2269, Ornaldo, FI., July 16-18, 1990.

[20] W. G. Brady and G. Ludwig, "Theoretical and experimental studies of impinging jets," IAS 31st Annual Meeting, IAS paper No. 63-29, NewYork, Jan. 21-23, 1963.

NOMENCLATURE

C_p	pressure coefficient, $=(p-p_\infty)/0.5\rho_\infty V_\infty^2$
$C_{p\max}$	maximum C_p on the ground plane
$C_{p\min}$	minimum C_p on the ground plane
C_1, C_2	Standard turbulence coefficient
C_D, C_μ	turbulence model constants
$C_{1\varepsilon}, C_{2\varepsilon}, \sigma_\varepsilon$	turbulence model constants
d	diameter of the nozzle
G	turbulence generation
h	perpendicular height of nozzle exit above ground
k	turbulent kinetic energy
ℓ	horizontal distance between the nozzle and the intake
m	mass flow rate
p	pressure
Pr	nozzle pressure ratio p_o / p_∞
Re	Reynolds number
S_ϕ	source of ϕ per unit volume
t	time
T	***temperature***
u	local velocity
u'	turbulent fluctuation velocity
U_τ	friction velocity
U^+	dimensionless near wall velocity
$\vec{V}$	velocity in vector form
V_e	effective velocity ratio, $=\sqrt{(0.5\rho_j V_j^2)/(0.5\rho_\infty V_\infty^2)}$
V_j	jet exit velocity
V_∞	freestream (cross-flow) velocity
x	distance measured against the cross-flow
y	vertical distance measured along the free jet flow
y^+	dimensionless wall distance
ε	rate of dissipation of turbulent kinetic energy
ϕ	general variable ($\phi = 1, u, v, k, \varepsilon,$ or T)
Γ	turbulent exchange coefficient
$\sigma_\varepsilon, \sigma_k$	standard turbulent constant
σ_T	Prandtl number
μ	laminar dynamic viscosity
μ_t	turbulent (eddy) viscosity
ν	kinematic viscosity
ρ	density

subscripts

i	intake conditions
j	jet exit conditions
p	ground vortex maximum penetration point
ph	hot gases penetration
s	ground vortex separation point
v	ground vortex core point
∞	ambient (cross-flow)

BIOGRAPHY

B. A. Jubran, Professor and Head of The Department of Mechanical and Industrial Engineering at Sultan Qaboos University (1999-Present). Professor, Mechanical Engineering Department at the University of Jordan (1994-1997). Associate Professor (1989-1994). Assistant Professor (1984-1989). Head of Mechanical Engineering Department (1989-1991). He has taken position at the International Islamic University Malaysia (1997-1999) where he has served as a Deputy Dean (Academic Affairs) at the Faculty of Engineering and University Senate member (1997-1998). He has been a Visiting Professor at the University of Karlsruhe, Germany, Research Leave (Summer 1994). A Visiting Professor at the Royal Military College of Science (Cranfield University), Britain (1991-1992). He has also served as Director of the Arab Council for Training Students of Arab Universities, ACTSAU and National Secretary of the International Association for the Exchange of Students for Technical Experience (IAESTE), IAESTE-Jordan (1994-1997). He has worked as a project Engineer at Abu-Dhabi National Oil Company, UAE (1980-1981).