*19th ICAS Congress/AIAA Aircraft System Conference*
*Anaheim (CA), U.S.A., September 18-23, 1994*

ICAS-94-6.6.5

# A NUMERICAL SIMULATION OF SIDE JET FLOW IN A JET- ENGINE COMBUSTOR MODEL

*G. Esposito, F. Gamma, F. Stella*

*Dipartimento di Meccanica ed Aeronautica, Universita' degli studi di Roma "La Sapienza*

## ABSTRACT

In the present paper a numerical study of jet in a crossflow field is presented. Numerical test are presented for various type of jet gases (CO, $CO_2$, $CH_4$, He, $H_2$ and air), the main crossflow has been choose to be air.

Numerical simulations have been performed by means of finite volume discretisation, using a non staggered variable location, a second-order upwind scheme for the convective terms and the PISO algorithm embodied in STAR-CD numerical code.
For turbulence modelling, the k-ε approach has been used.

Numerical results have been validate by comparison with available experimental data obtained, using LDV technique, on the same combustor model.

## NOMENCLATURE

$k$ = turbulent kinetic energy
$m_c$ = mass fraction of constituent c
$s_i$ = source term
$u_j$ = velocity component
$D_c$ = molecular diffusivity of component c
$\Gamma_\phi$ = diffusion coefficient
$\varepsilon$ = dissipation velocity
$\mu_t$ = turbulent viscosity
$v_j$ = jet velocity
$v_f$ = crossflow velocity
$\rho$ = density
$d$ = orifice diameter
$\phi$ = dependent variable
$k$ = heat conductivity
$c_p$ = specific heat at constant pressure
$\sigma_{ct} = \left(c_p / \mu_t k\right)$, turbulent Schmidt number
$q = \left(\rho_j v_j^2 / \rho_f v_f^2\right)$, momentum flux ratio
$r = \left(\rho_j / \rho_f\right)$, density ratio

## INTRODUCTION

Combustion performance in a gas turbine plays a very important role in overall performance of aeronautics jet-engines. As matter of fact, in modern aircraft, around 3% of jet-engine fuel is unburned, mainly because of poor mixing and relative short residence time inside the main combustor.

From the other side it is very well-know that in a continuous combustion chamber, as a gas turbine combustor, a wide range of complex and interacting physical phenomena plays a relevant role. Same of this phenomena are: fuel spray atomisation and vaporisation, turbulent transport, finite rate chemistry of combustion, etc. etc. For these reason design and analysis of gas turbine combustion chamber are a formidable challenge in terms of thermochemistery and fluid dynamics models.

It is well-know that the study of the behaviour of jet injection into a subsonic crossflow is one more sensitive point for the evolution of combustion efficiency. Thus is very important to identify the interaction flow field and the passive

scalar distribution (for example the concentration) in the primary combustion zone, where there is fuel injection in the air crossflow.

It is therefore very important to know one or more parameters able to represent the jet mixing and penetration in the crossflow, in spite of combustion chamber geometry.

In a previous paper Holdeman[1] has found using an experimental technique that one of the most important parameters for the prediction of jet penetration is the momentum flux ratio (q) between the main flow and the jet. In his experiment, Holdeman, also found that the deep of penetration of the jet into the main flow is insensitive to density of the fluid injected for several type of gases, for a wide range of molecular weight. Only for very low density gases, as $H_2$, density plays an important role strongly decreasing the deep of jet penetration

In a successive paper Andriani et alii[2] studied, using a LDV technique, the interaction between jet and crossflow. Following the results obtained by Holdman they used the momentum flux ratio (q) as main parameter, finding a good agreement with the results obtained by Holdman

In the present paper a direct calculation of the flow field resulting from the interaction of gas jets in a air crossflow is presented. Several different gases , such as CO, $CO_2$, $CH_4$, He, $H_2$ and air, have been considered as jet fluid.

Furthermore the evaluation of the influence of some typical flow parameters on jet penetration, such as flux momentum ratio (q) and density ratio (r), is presented.

The actual numerical calculations have been carried out using the numerical code STAR-CD. STAR-CD uses a k-ε approach for modelling turbulence embodied in a finite volume discretisation technique. In the present paper we adopted a modified version of the standard k-ε model, as proposed by Launder and Spalding[3] for the study of internal flow.

In the study of jet mixing problems in combustor chambers, k-ε model of turbulence has been previously successfully used by several authors. See for example Shyy et alii[4] and Lee et alii[5]. In particular Lee et alii have shown that for the type of flows under study several variant of the turbulence model, such as k-ε , ASM and RSM, give similar results.

In the following of the paper, numerical test for the accuracy of the numerical code and turbulence model adapted, a comparison with the experiment findings by Andriani et alii[2] will be also presented.

## EXPERIMENTAL AND NUMERICAL CONFIGURATION

The physical problem considered is the study of a 3D jet into a crossflow. The geometrical domain is a vertical duct (100 mm X 100 mm) with a lateral orifice (d=3 mm) (se fig. 1)

The crossflow is confined in the vertical duct and the jet is injected 10 mm downstream the main inlet section. Various type of fluid have been considered as jets , while we always used air for the main flow.

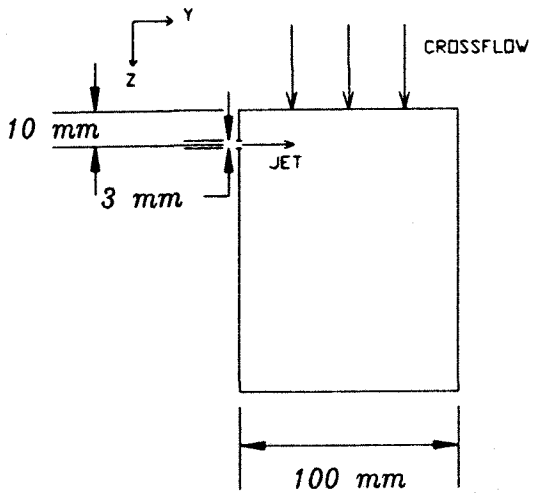


**Fig. 1**

The momentum flux ratio (q) between the main flow and the jet has been used as main parameter as well several different fluids have been used as jet, in order to study the importance of molecular weight in the interaction between jet and crossflow.

The fluids used for the jet are: $CO_2$, CO, $CH_4$, He, $H_2$ and air, resulting a variation of molecular weight between 2 ( $H_2$ ) and 44 ( $CO_2$ ).

The momentum flux ratio (q), definite as

$$q = \frac{\rho_j V_j^2}{\rho_f W_f^2} \qquad (1)$$

has been keep constant and equal to 28.7. Only in few test q has been changed in 15.6 and 42.6.

The accuracy of numerical and turbulence models has been first evaluated by comparing results obtained by Andriani et alii[2]

## NUMERICAL APPROACH

### DISCRETE FORMULATION

The differential equation governing the conservation of mass, momentum, energy etc. within the fluid are discritised using a finite volume

technique[6][7]. Thus, they are first integrated over the individual computational cells and then approximated in terms of the cell-centred nodes values of the dependent variables. This approach has the merit, among others, of ensuring that the discretised forms preserve the conservation properties of the parent differential equations.

For seek of brevity it is convenient to write the field equations using a general vectorial form:

$$\frac{\partial}{\partial t}(\rho\phi) + \nabla\cdot\left(\rho\bar{u}_\phi\phi - \Gamma_\phi\nabla\phi\right) = s_\phi \qquad (2)$$

$\phi$ stands for one of dependent variables ( i.e. $u_i$, k, h, etc.), $\Gamma_\phi$ and $s_\phi$ are the associated "diffusion" and "source", coefficients, which can be deduced from the parent equations.

The solution procedure is based on the well know PISO[8] method, and the implicit discretisation of time derivatives has been used, resulting in a great stability to large time step.

The actual numerical calculations has been carried out using the numerical code STAR-CD. Further details on numerical code and discrete formulation can be found in STAR-CD user's manual[9].

## MULTICOMPONENT FLOW EQUATIONS

One of the main problem when dealing whit multicomponent fluid flow is the evaluation of the local concentration of different species.

As already noted, the solution of the scalar transport equations for the turbulence parameters, enthalpy and chemical species conservation take place within the sequential operations performed during each iteration. Calculation of thermophysical properties, such as density and viscosity, is also included in this sequence. The inter linkages between these additional variables and the flow field are handled by linearisation using an appropriate factorisation within PISO sequence.

The local mass concentration of each constituent c of the fluid mixture may be expressed as a mass fraction:

$$m_c = \frac{M_c}{M_{tot}} \qquad (3)$$

where obviously:

$$\sum_c m_c = 1 \qquad (4)$$

Each constituent c of the local mixture is assumed to be governed by a species conservation equation of the form:

$$\frac{\partial}{\partial x_j}\left(\rho\tilde{u}_j m_c - F_{c,j}\right) = s_c \qquad (5)$$

where the diffusion flux components for turbulent flow are:

$$F_{c,j} = \rho D_c \frac{\partial m_c}{\partial x_j} - \overline{\rho u_j' m_c'} \qquad (6)$$

where $D_c$ is the molecular diffusivity of component c and

$$\overline{\rho u_j' m_c'} = -\frac{\mu_t}{\sigma_{c,t}}\frac{\partial m_c}{\partial x_j} \qquad (7)$$

are the Reynolds stresses that depend from the turbulent Schmidt number of each single species.

But in reason of the fact that we considered the turbulence as isotropic and the Reynolds numbers studied is sufficiently large , we can use in eq. (7) the molecular Schmidt number instead of the turbulent Schmidt number, as also stated by Hinze[10].

To calculate the local mixture viscosity we have used as mass-weight averages of the component values, i.e.

$$\phi = \sum_{c\neq bg} m_c\phi_c + \left(1 - \sum_{c\neq bg} m_c\right)\phi_{bg} \qquad (8)$$

here $\phi$ is the mixture value of the aforementioned property, $\phi_c$ is the property value of constituent c and $\phi_{bg}$ is the property value of the background fluid.

## TURBULENCE MODEL

In terms of turbulence model, measurements of the flow and turbulence transport characteristics of gas turbine combustor have been reported by Heitor and Whitelaw[11] . According to them a scalar effective viscosity turbulence model (e.g. the k-$\varepsilon$ model) should be adequate for the flow in the dilution section of the combustor since the production of turbulent kinetic energy is largely caused by the interaction of shear stress with shear strain.

In the upstream recirculation zone, however,

the flow is characterised by a large mean radial pressure gradient. Since the production of turbulent kinetic energy is not accounted for by mean gradient mechanisms, an eddy-viscosity model would under-predict the turbulent energy levels, and is unlikely to provide correct trends for the turbulent transport of scalar quantities.

Anyway at present we still use a k-ε turbulence model, along whit the wall function treatment for the near-wall region, also for complicated three-dimensional flows in the gas turbine combustor, because it appears to be the best compromise between accuracy and computational cost.

A number of modifications have been proposed in order to overcome these deficiencies, including changing the values of the constants of the standard model as recommended by Launder and Spalding[3].

For the specific case of internal combustor chambers Lee et alii[5] have tested several variant of the turbulence model, such as k-ε , ASM, RMS. The comparison between the three different models gives similar results, so that we consider reasonable the choose of k-ε modified using Spalding and Launder coefficients.

## *COMPUTATIONAL GRID*

A tridimensional mesh has been created in order to carry out numerical calculations, using a right-handed cartesian co-ordinates so that the positive direction of the Z-axis is the same of the cross-flow, and the y-axis is directed as the jet-axis.

The mesh has been refined around the jet of the axis, and a circular reference has been used to simulate the circular orifice where is the jet injection (see fig. 2), since all experimental results used to compare numerical data refer to a jet from a circular orifice.

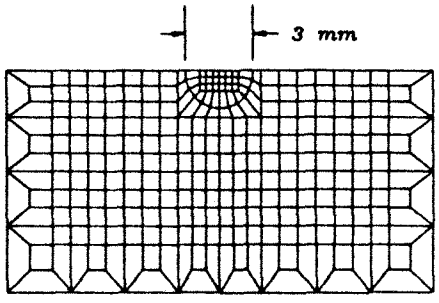


**Fig. 2**

The characteristic dimensicn of the mesh near the centre of the jet is 0.006 d (d=3 mm is the diameter of the orifice of the jet).

According to experimental results the plane which is parallel to the principal flow direction and which goes through the jet axis, has been assumed to be a symmetry plane.

More than fifty thousand cells have been used for the numerical calculations.

## DISCUSSION OF RESULTS

Numerical results have been compared first with some experiments in order to validate and estimate the accuracy of the numerical code adopted.

As main parameter the jet centerline has been plotted as a function of the horizontal abscissa (z).The jet centerline has been defined as the position where, at each horizontal section z, the velocity profile is maximum.

As inlet condition for the crossflow the velocity profile and level of RMS obtained experimentally by Andriani et alii[12] has been used.

Although the maximum of error on the centerline position between numerical and experimental results, is 0.5 mm, that is relatively large when compared with the orifice jet diameter of 3 mm, the results obtained compare very well with experimental ones (see fig. 3, 4). As matter of fact in most of the points numerical results are compatible with experimental error, estimated by Andriani et alii[12], we have considered this results accurate enough for the goals of the present paper.

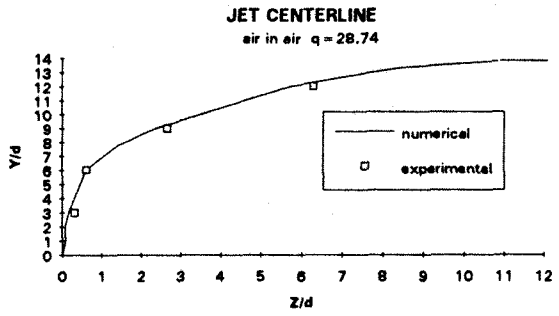


**Fig. 3**

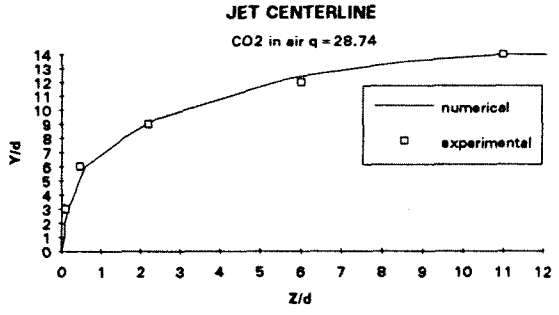


**Fig. 4**

Extensive numerical investigation has been conducted for q=28.7, using different type of gasses so that density has been varied between 0.08 (hydrogen) and 1.79 ( carbon dioxide ). Results are discussed as function of density ratio ($r = \rho_j/\rho_{air}$) between the jet density and the crossflow one ($\rho_{air}$=1.205).

For jets of carbon dioxide (molecular weight=44, r=1.49), carbon monoxide ($m_w$=28, r=0.94) and air, in which the density ratio varies from 0.95 to 1.5, it is possible to note that the centerlines are very similar (see fig 5.); for the carbon monoxide jet and the air jet, not only the graphics are superimposed but also numerical results are the same within the third digit.

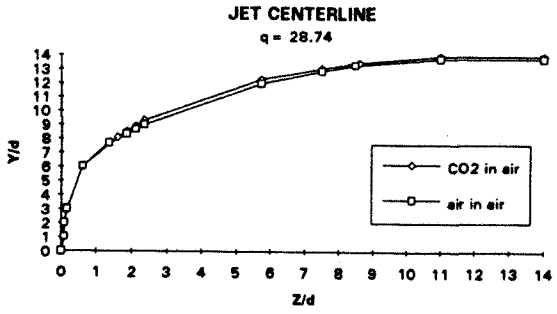


Fig. 5

The methane ($m_w$=16, r=0.55) jet centerline graphic shows a lower penetration with respect to the air jet centerline (see fig. 6).

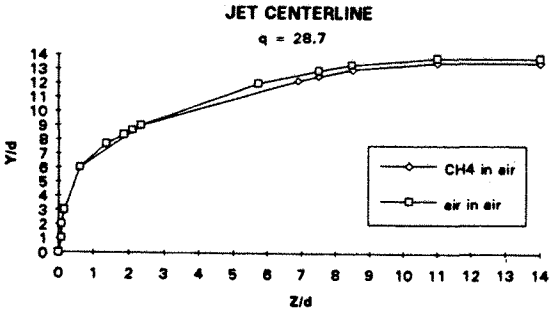


Fig. 6

In the numerical test conducted with helium ($m_w$=.4, r=0.13) and hydrogen ($m_w$=2, r=0.06) as jet fluid, although the difference in terms of density ratio is small it is possible to note a strong difference between the numerical curves (see fig 7.).

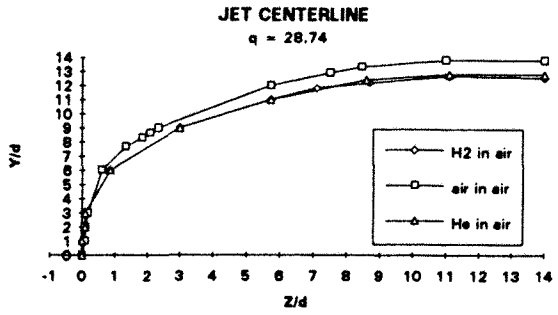


Fig. 7

These result shows that penetration is sensitive to density ratio when this parameter is small enough.

On the contrary we can see that there is a wide range of r where penetration is nearly constant.

In fig. 8 the asymptotic value of centerline versus the density ratio has been plotted. It is evident that gases with higher molecular weight have a stronger penetration into the crossflow. It can also be seen in the diagram for high value of r that the curve is nearly flat showing a weak dependence of penetration from molecular weight. On the contrary for low density gasses the curve is very steep, showing a strong dependence from this quantity.

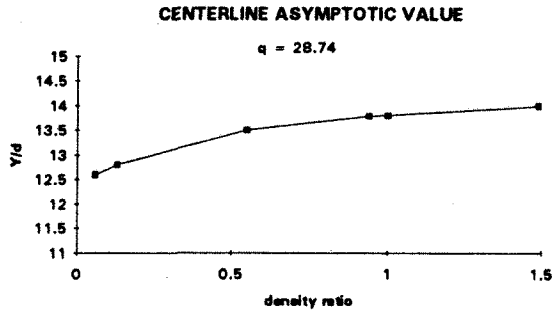


Fig. 8

The fig. 9 shows the position on z axis where centerline reaches 95% asymptotic position as function of the density ratio . This position moves downstream across the main flow the molecular weight increases. This is consequence of the stronger penetration obtained with larger r (i.e. 3 mm).

Always it is important to notice that 95% of the asymptotic position for the centerline is reached after no more than 8.7 diameters (i.e. 26 mm in our case), and the difference between the value obtained using hydrogen and carbon dioxide is less then one diameter.

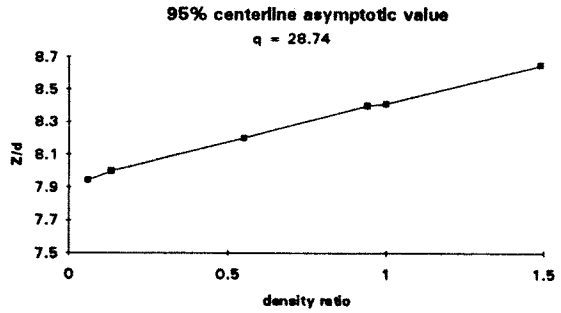


**Fig. 9**

In fig. 10, the results obtained simulating a jet of air in air crossflow are shown. In this case the momentum flux ratio (q) has been varied from 18.6 up to 42.6, meanwhile the density ratio is obviously constant and equal to one. It is evident that the penetration is strongly modified by the variation of q.

We have chosen a percentage variation of q similar to the percentage variation of r that we have between air and methane and air and dioxide carbon. In that cases (see fig. 5 6) the variations of penetration was, as discussed before, negligible.

This clearly indicates that q is a good parameter for representing the penetration behaviour.

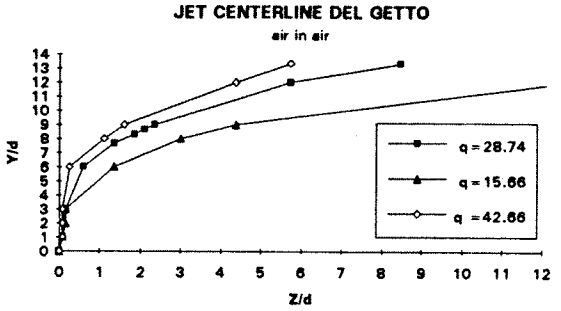


**Fig. 10**

Acknowledgement: Computational Dynamics Ltd. is gratefully acknowledged for the use of STAR-CD.

## REFERENCE


1. Holdeman J. D: , "Mixing of Multiple Jets With a Confined Subsonic Crossflow",*J. Prog. Energy Combust. Sci.*, Vol. 19, 1993.
2. Andriani R., Coghe A., Brunello G., "Analisi Sperimentale del Mescolamento di un Getto Gassoso Immesso in una Corrente Trasversale", *46° Congresso ATI, Gaeta (IT), 25-27 settembre 1991.*
3. Launder B: E: and Spalding D. B., "The Numerical Computation of Turbulent Flows", *Computer Methods in Appl. Mech. and Engin.*, Vol 3, 1974
4. Shyy W., Correa S: M., Braaten M. E., "Computation of Flow in a Gas Turbine Combustor", *Combust Sci. and Tech.*, Vol. 58, 1988
5. Lee D., Yeh C. L., Tsuei Y. M., Chou J., "Numerical Simulations of Gas Turbine Combustor Flows", *J. of Propulsion and Power,* Vol 9, No 2, 1993
6. Gosman A. D., Pun W: M., Runchal, A. K., Spalding D. B., "Heat and Mass Transfer in Reicirculatig Flows", Accademic Press, 1969
7. Patankar, S. V., "Numerical Heat Transfer and Fluid Flow", Hemisphere, 1980
8. Issa, R.I., "Solution of the Implicity Discretised Fluid Flow Equations by Operator-Splitting", *J. Comp. Phys.*, 62,1986
9. "Star-CD Version 2.2 Methodology", Computational Dynamics Ltd., 1993
10. Hinze J. O., "Turbulence",McGraw-Hill, 1975
11. Heitor M. V., Whitelow J. H., "Velocity Temperature and Species Characteristics of the Flow in a Gas Turbine Combustor", *Combust.and Flame,* 64 1,1986
12. Andriani R., Coghe A., Brunello G., "private communication"
13. Lilley, D.G., "Flowfield Modelling in Pratical Combustors: a Review", *J. of Energy,* July-august 1979
14. Shyy W., Braaten M. E., Burrus D. L., "Study of Three-Dimensional Gas Turbine Combustir Flow", *Int. J. Heat Mass Transfer,* Vol. 32, 1989.
15. Esposito G., Gamma F., "Analisi Numerica dall' Interazione Fluidodinamica di Getti Gassosi", *48° congresso ATI, Taormina (IT), 28 sett.-1 ott., 1993*
16. Roth K. R., Fearn R. L., Thakur S. S., "Evaluation of a Navier-Stokes Prediction of a Jet in a Crossflow", *J. of Aircraft,* Vol. 29, No 2, 1992