ARTICLE NO. 79-1194R

C80-125

Development of a Gas Turbine Combustor Dilution Zone Design Analysis

20007
60001
60002

A. S. Novick,* J. R. Arvin,† and R. E. Quinn‡
Detroit Diesel Allison Div., General Motors Corp., Indianapolis, Ind.

Recent advancements in developing an analytical technique applicable to gas turbine combustion design problems is presented. A three-dimensional mathematical model defines the trajectories of a system of dilution jets within a combustor dilution zone and transition section. Interactions of jets with each other and confining walls are included. The analytical simulation and solution techniques are detailed due to the uniqueness of modeling the multijet trajectories and interactions in a confined environment. Comparisons of the analytical predictions with available experimental data are shown. Results illustrating practical combustor flow problems show that progress has been made toward the realization of a useful combustor design capability.

Nomenclature

A	= dilution hole area
a	= ellipse ratio
C_D	= discharge coefficient
D	= combustor diameter
d	= dilution hole diameter
H	= total flux of enthalpy
h	= stagnation enthalpy
J	= total number of jets
$\bar{N}$	= normal to a surface
P	= pressure
r	= radius
S	= distance along jet centerline
SL1-6	= streamline
SS1-6	= stream surface
T	= temperature
$\bar{V}$	= velocity
W	= mass flow
X	= total momentum flux
α	= spreading rate constant angle defined in Fig. 5
η	= nondimensional radial coordinate
γ	= angle defined in Fig. 3
ν	= angle defined in Fig. 4
ρ	= density
σ	= jet lateral width

Subscripts

c	= center line
g	= hot gas
i,j	= jet
o	= origin, potential core

I. Introduction

MIXING of cool dilution air with hot primary stream gas is a design area which receives considerable attention during the design and development of a contemporary gas turbine engine combustion system. The significance of this mixing process is manifested in engine life. High-temperature streaks in a combustor exit or a skewed radial temperature profile (hub-oriented) are detrimental to turbine vanes and blades, respectively. It is recognized that dilution mixing is a prominent problem within the gas turbine industry today; however, it will become an increasingly more difficult problem in the future with the advent of advanced technology, high-temperature-rise combustors. High temperature-rise combustion systems, out of necessity, minimize dilution air in order to have adequate air for combustion and liner cooling. Thus, combustor design engineers will require advances in analytical techniques that aid in the optimization of dilution mixing for engine hot-section life.

Present design practices rely on extensive rig and engine development testing in order to adjust the combustor exit temperature pattern to meet the demands of engine hot-section durability. This development testing is guided by empirical design analyses coupled with judicious experience of design engineers. Empirical design techniques have, for the most part, been developed from correlations of engine and rig data and experimentation on jets penetrating into an idealized environment. Semi-infinite, or in some cases, two-dimensionally confined isothermal or heated crossflow experiments have been used to determine single or multiple jet trajectories.[1-9] These experiments, in conjunction with analytical studies, have led to techniques for predicting jet trajectories and, in some instances, jet spreading and mixing. However, the design techniques which have evolved from these analyses and rig development test are becoming inadequate when presented with the task of extrapolating experience for more stringent dilution/mixing requirements.

The principal role of this dilution zone analysis is to serve as an engineering tool to assist in combustor development. Since vane as well as blade hot spots are of interest, the analysis must account for the three-dimensional geometry of a combustor and its associated transition section. In the analytical method, an approximate differential equation is solved to predict the velocity centerline trajectories of the multiple jets within the dilution zone. The theoretical formulation includes the interactions of jets with each other and with confining combustor walls. The velocity centerline trajectories of the dilution jets in the three-dimensional combustor geometry, three-dimensional space curves, provide the design engineer with the capability to interrogate a dilution zone design with regard to the effects of hole location and mass flow.

Presented as Paper 79-1194 at the AIAA/SAE/ASME 15th Joint Propulsion Conference, Las Vegas, Nev., June 18-20, 1979; submitted July 9, 1979; revision received Feb. 1, 1980.

Index categories: Combustion and Combustor Designs; Jets, Wakes, and Viscid-Inviscid Flow Interactions; Airbreathing Propulsion.

*Research Scientist, Combustor Research & Development. Member AIAA.

†Supervisor, Turbine Design.

‡Senior Research Engineer, Mechanics Research.

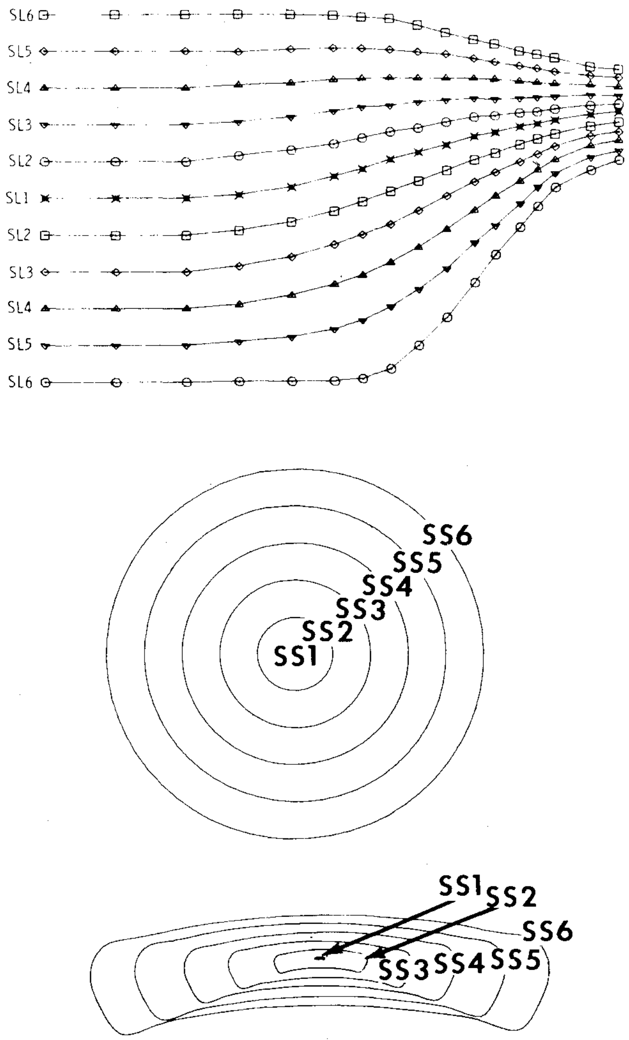

Fig. 1 Contemporary can combustor dilution mixing region.

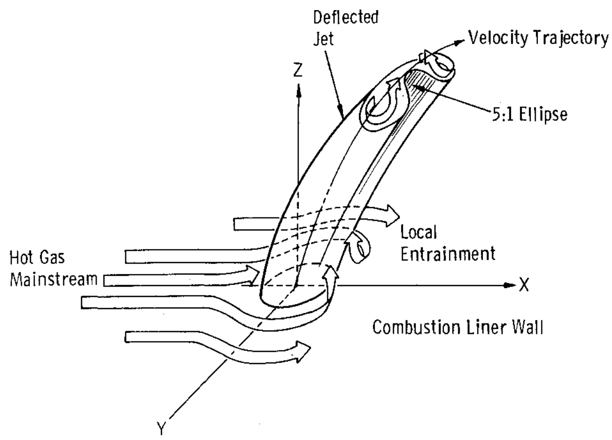

Fig. 2 Schematic of jet crossflow interaction.

II. Simulation and Solution

A. Physical Problem

The problem to be solved is that of describing the phenomena of cool interacting dilution jets penetrating into a hot, nonuniform, crossflowing, confined combustor environment. The present work describes an analytical technique which will allow combustor designers to predict the path of all dilution jets from their point of entry into a combustor to the exit plane of the transition section. The analytical technique includes three-dimensional geometry, a streamtube representation of the hot gas within a dilution zone and transition section, and a mass/momentum balance formulation for calculating the trajectories of the dilution jets.

Flow streamlines within the dilution zone and transition section of a contemporary can combustor, typical of one from a can-annular combustion system, are shown in Fig. 1. The streamlines (top sketch) are determined using a radial equilibrium analysis[10] to analyze the flow in a plane which passes through the center of the combustor and the engine axis, i.e., a meridional plane. Herein the hot-gas flowfield is that which would exist if the flow were axisymmetric about the engine centerline and the dilution jets not present. The outermost streamlines, SL6, represent the combustor wall and thus the figure illustrates the dilution-to-transition flowpath convergence in the axial direction. The center and bottom sketches illustrate the plane of the dilution holes and combustor exit, respectively. The "stream surfaces" are generated by revolving the streamlines around SL1 while maintaining the same percent spacing as depicted in the top sketch.

This geometry is typical of a dilution mixing region in most gas turbine combustors. The mainstream flow velocity and pressure are determined from the local streamline orientation and the convergence/divergence of streamtube elements. Inherent in the freestream velocity determination is the assumption that dilution jets affect only the magnitude of the velocity and not its direction. The velocity magnitude is assumed to be influenced by the gross blockage of the mainstream by the jets. The calculation procedure for these mainstream parameters is straightforward but tedious, and will not be part of this paper. It suffices to mention that a significant amount of computer code is involved to locate the streamtube of interest and calculate the local flow conditions, i.e., velocity and pressure.

Boundary and initial conditions are required to complete the analytical specification of the problem. The boundary condition applicable to the mainstream is that the flow must be tangent to the liner wall (the conditions pertinent to the jets will be discussed in Sec. IIC). In order to predict the jet trajectories, a description of the circumferential and radial distributions of velocity and temperature in the hot primary inlet flowfield at the plane of the dilution holes is necessary. There are some techniques available with which the combustor designer can qualitatively determine these parameters.[11] However, quantitative primary zone flow descriptions are not yet available.

The jet initial conditions are determined from a quasi-three-dimensional combustor airflow management analysis similar to the one proposed by Adkins.[12] Parameters which must be specified are the jet momentum (liner pressure drop), entry angle, and temperature. Since in practical situations the annulus flow for can combustors is circumferentially distorted, it is necessary to have prior knowledge of the conditions for each dilution port in order to take full advantage of the dilution zone design analysis. This is not a significant problem for annular combustors, and an axial airflow analysis is adequate.

B. Jet Deflection by Crossflow

The analytical development of the three-dimensional jet trajectory model is formulated analogous to the circular jet in a crossflow analysis developed by Braun and McAllister[13] and utilized in Ref. 9. However, in addition to the three dimensionality, the present analysis is unique in that it includes a variation of the rate of lateral spreading of a jet with the angular orientation between the jet and the crossflow and provides for the crossflow to have an arbitrary variation of velocity and angular orientation with respect to the jet. Due to these generalized crossflow considerations, the resulting velocity centerline trajectories are three-dimensional space curves. A sketch of a jet being deflected by a crossflow is shown in Fig. 2.

Inherent in the analysis are the following assumptions:

1) Mass entrained by a jet carries its full complement of freestream momentum into the jet.

2) Flow velocity within a jet is parallel to the direction of the centerline trajectory.

3) Jet lateral spreading is a linear function of the distance from an apparent origin.

4) Jet shape is elliptical with a 5:1 major to minor axis ratio.

5) The major axis of the elliptical cross section is perpendicular to the local directions of the crossflow velocity and the jet centerline trajectory. The minor axis is perpendicular to the centerline trajectory in the local plane of principal curvature.

6) Pressures on the jet surface are imposed by the crossflow and are independent of the jet/crossflow interaction.

7) Forces acting on a jet element are due to changes in momentum flux and pressure.

In the solution technique, a stepwise integration method is used to calculate the differential change in jet orientation $\Delta\gamma$ as a function of a change in arclength along the trajectory, $|\overline{\Delta S}|$. Consequently, the trajectory is a buildup of differential elements whose flow direction is constant.

Lateral entrainment width of a jet is assumed to be governed by the relationship

$$\sigma = \sigma_o + S(k_1 \sin\gamma + k_2 \cos\gamma) \tag{1}$$

where k_1, k_2 are proportionality constants.

The arclength S of a jet may be expressed as the summation of the n incremental displacements,

$$S = \sum_{i=1}^{n} |\overline{\Delta S_i}| \tag{2}$$

The growth in lateral width is used to calculate entrainment of the hot gas.

The inclusion of two constants in Eq. (1) is to simulate the rapid entrainment near the orifice for normal jets and the approach the submerged jet case asymptotically as the jet becomes nearly parallel in the flowfield downstream of the orifice. When the jet and crossflow are approximately parallel, the growth of the entrainment width is dominated by the cosine term. In accordance with the classical theory of a submerged jet,[14] the coefficient of this term, k_2, is taken to be 0.32. A value of k_1 equal to 0.7 was determined from a best fit of experimental data in order to agree with the high initial entrainment rates associated with the component of crossflow normal to the jet. In conjunction with the model developed in Sec. IIC, this two-coefficient formulation provides the capability to calculate jet trajectories over the range of momentum ratios pertinent to gas turbine combustors.

The equation to calculate the deflection of a jet by the crossflow is derived from a balance of momentum flux normal to the jet axis. For the finite control volume depicted in Fig. 3, a momentum balance leads to the following equation

$$\tan(\Delta\gamma) = \frac{\sigma |\overline{\Delta S}| [P_4 - P_5) + \rho_g |\bar{V}_g|^2 \sin^2\gamma}{X_j + \rho_g |\bar{V}_g|^2 \sigma |\overline{\Delta S}| \sin\gamma\cos\gamma + \dfrac{\pi}{4a}(P_1 - P_3)\sigma^2} \tag{3}$$

where, as shown in Fig. 3, P_2 is the pressure at the jet center, and P_1, P_3, P_4, P_5 are the pressures at the nodal points. This equation is solved in a repetitive manner along the trajectory to obtain changes in the jet direction in its plane of principle curvature.

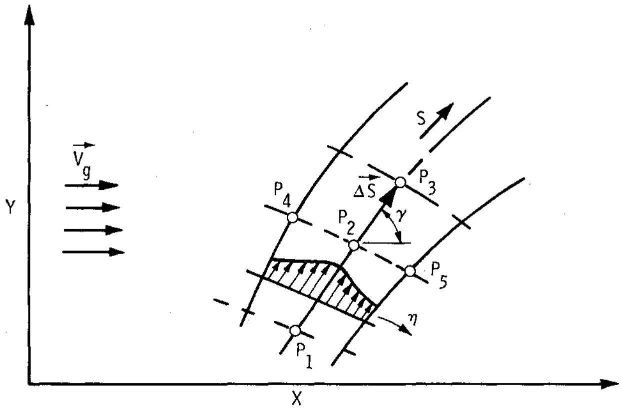

Fig. 3 Control volume in plane of jet trajectory.

The deflection of the jet in the binormal direction $\bar{N}_1$ is due to the static pressure gradients in the crossflow,

$$\sin|\Delta\nu| = (P_6 - P_7)\frac{|\overline{\Delta S}|\sigma}{aX_j} \tag{4}$$

where, as shown in Fig. 4, $\Delta\nu$ is the incremental angular deflection of $\overline{\Delta S}$ in the direction of $\bar{N}_1$, and P_6, P_7 are the pressures at the nodal points.

The momentum flux X_j parallel to the jet path is computed from the summation,

$$(X_j)_n = (X_j)_o + \sum_{i=1}^{n} \rho_g |\bar{V}_g|^2 \sigma |\overline{\Delta S_i}| \sin\gamma \tag{5}$$

where $(X_j)_o$ is the initial jet momentum flux at the orifice discharge defined as

$$(X_j)_o = C_D \rho_g |\bar{V}_g|^2 A_j \tag{6}$$

The summation on the right side of Eq. (5) defines the total momentum entrained into the jet.

The changes $\Delta\gamma$ and $\Delta\nu$ in the angles γ and ν are used to update the local orientation of the centerline trajectory with respect to an overall system of Cartesian space coordinates. Spatial orientation of the major axis of the jet cross section is given by the components of a unit vector

$$\bar{N}_1 = \frac{\bar{V}_g \times \overline{\Delta S}}{|\bar{V}_g|\,|\overline{\Delta S}| \sin\gamma} \tag{7}$$

This vector cross-product yields a direction which is perpendicular to the tangent to the centerline trajectory and to the velocity vector of the crossflow. The angle γ is computed from the vector dot product relationship

$$\gamma = \cos^{-1}\left[\frac{\bar{V}_g \cdot \overline{\Delta S}}{|\bar{V}_g|\,|\overline{\Delta S}|}\right] \tag{8}$$

Specification of the plane of the jet cross section is completed by defining the spatial direction of the minor axis of the ellipse. Since it is assumed to be perpendicular to both the jet trajectory and the major axis, its direction is given by the components of the unit vector

$$\bar{N}_2 = \frac{\bar{N}_1 \times \overline{\Delta S}|}{|\bar{N}_1 \times \overline{\Delta S}|} \tag{9}$$

The Cartesian components of vectors $\bar{N}_1$ and $\bar{N}_2$ are updated repeatedly as the solution for the jet centerline trajectory is propagated along the beamwise steps $|\overline{\Delta S}|$. The vector $\overline{\Delta S}_n$ is updated by the equation

$$\overline{\Delta S}_{n+1} = \frac{\overline{\Delta S}_n + |\overline{\Delta S}_n|(\tan(\Delta\gamma)\bar{N}_2 + \tan(\Delta\nu)\bar{N}_1)}{\sqrt{1+\tan^2(\Delta\gamma)+\tan^2(\Delta\nu)}} \tag{10}$$

Lateral spreading and decay of the maximum value of jet velocity along the centerline trajectory are modeled by an assumed empirical velocity distribution. The form of this distribution is

$$\frac{|\bar{V}_j| - |\bar{V}_g|}{|(\bar{V}_j)_c| - |\bar{V}_g|} = (1+0.414\eta^2)^{-2} \tag{11}$$

where $(\bar{V}_j)_c$ is the maximum jet velocity along the centerline trajectory. The coordinate η refers to a local polar coordinate system (r,ϕ) where

$$\eta = r/r_{1/2} \tag{12}$$

The half-radius $r_{1/2}$ of the jet (the radius at which the velocity is equal to one half the maximum value $|\bar{V}_j|_c$) varies with the polar angle ϕ such that level surfaces of jet velocity correspond to 5:1 ellipses. The variation of the half-radius is given by

$$r_{1/2} = \alpha S\sqrt{1-k^2\cos^2\phi} \tag{13}$$

where the spreading rate constant α and the eccentricity k have to be determined experimentally (for a 5:1 ellipse $k^2=0.96$).

The maximum velocity is assumed to decay with distance along the jet path by the empirical relation

$$\frac{|(\bar{V}_j)_c| - |\bar{V}_g|}{|(\bar{V}_j)_o| - |\bar{V}_g|} = \frac{C_1}{S/d_j} \tag{14}$$

C_1 is a constant determined by normalization of the velocity profile. The normalization of the profile is based on conservation of momentum for all the jets and the crossflow.

In addition to the total momentum flux X_j of the jet, additional total fluxes of mass and energy are computed for the purpose of normalization of the velocity and temperature profiles. The total mass flux of the jet is

$$W_j = \rho_{jo}|(\bar{V}_j)_o|A_j + \sum_{i=1}^{n} \rho_g |\bar{V}_g| \sigma |\overline{\Delta S}_i| \sin\gamma \tag{15}$$

The total flux of stagnation enthalpy of the jet is

$$H_j = \rho_{jo}|(\bar{V}_j)_o|A_j(h)_{jo} + \sum_{i=1}^{n} \rho_g |\bar{V}_g| \sigma |\overline{\Delta S}_i| \sin\gamma (h)_g \tag{16}$$

C. Jet Interactions

Practical combustors utilize a multiple-jet dilution zone to promote uniform mixing within a small mixing region. In order to develop a realistic simulation applicable to dilution zone design and development, one must include the trajectories of multiple jets and their interactions. The previously described jet trajectory analysis includes only the effect that the mainstream has on a jet, exclusive of any interactions. Therefore, supplemental conditions must be imposed on these trajectory equations to represent the forces that jets impose on each other.

Supplemental conditions are formulated with the aim of modifying the analytically predicted jet trajectory. The interaction model is based on the hypothesis that jets will interact within a half-radius of each other and subsequently become parallel. This hypothesis is based on observations of confined jets and replacing the confining wall with an equal and opposite jet.[2] At this stage in the development of the dilution zone design model, the entrainment of one jet by another is not included.

To calculate the influence that one jet has on another requires that the vector representation for each jet be resolved into the components shown in Fig. 5. All the jets must be at the same axial location in order to resolve them into components. An expression has been derived from an empirical fit of trajectory data[2] near a wall over a range of momentum

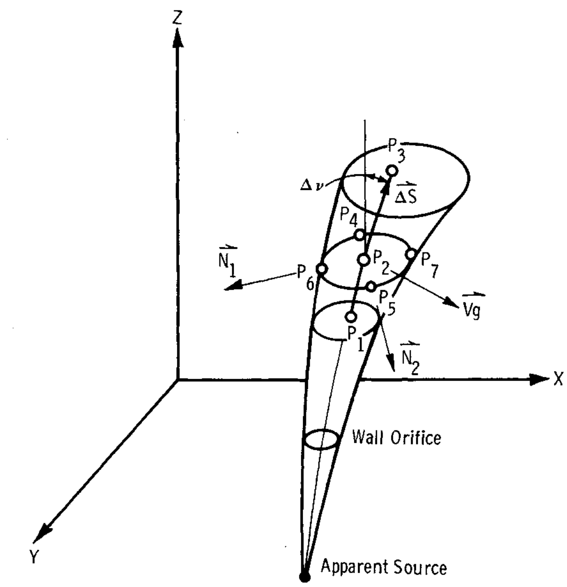

Fig. 4 Spatial orientation of jet cross section.

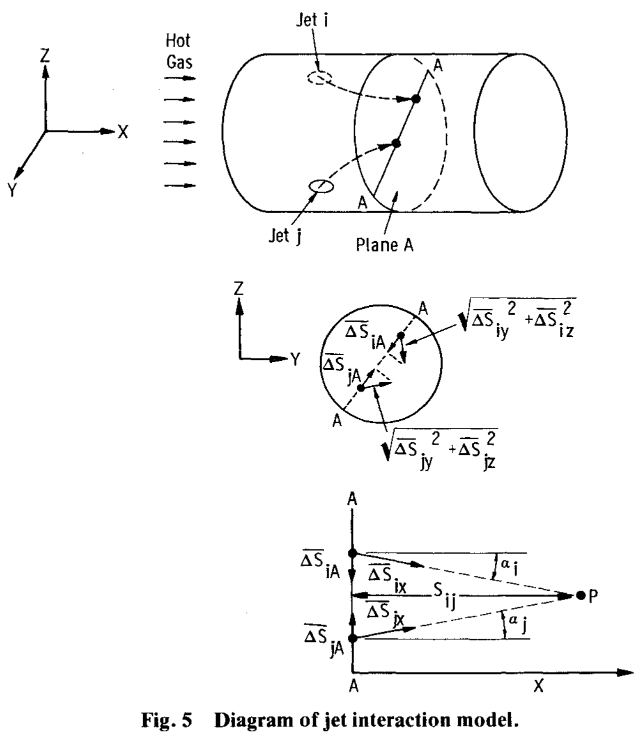

Fig. 5 Diagram of jet interaction model.

ratios for the change in orientation of the jth jet α_j as a function of its anticipated interaction

$$\Delta\alpha_j = \sum_{i=1}^{J} (|\alpha_i| + |\alpha_j|)\left[1 - \frac{S_{ij}}{S_{ij} + r_{1/2}}\right]^B$$

$$\times \frac{\rho_j |\bar{V}_j|^2 A_j}{\rho_i |\bar{V}_i|^2 A_i + \rho_j |\bar{V}_j|^2 A_j}$$

$$B = \exp\left(10.7\frac{\rho_g |\bar{V}_g|^2}{\rho_j |\bar{V}_j|^2} - 3.2\right) \tag{17}$$

The length S_{ij} shown in Fig. 5 is the distance that the jth jet will traverse along a straight line before colliding with the ith jet. The change in orientation of the jth jet reflects the resultant forces of the J-1 jets acting on it ($i \neq j$). The equation highlights the facts that high-momentum jets will dominate low-momentum jets and the further away a jet is from its opponent, in relationship to its size, the less of an effect it has on it.

The dilution zone design analysis utilizes Eq. (17) through redefining of the unit vector ΔS_i based on the new orientation α_i. In redefining the unit vector, the magnitude of its axial component remains constant. Thus one is able to calculate the magnitude of the remaining vector components while holding the resultant direction cosines constant.

A second interaction which needs to be included in a confined jet analysis is a jet colliding with a wall. In modeling this interaction, we utilize the above hypothesis that if a jet interacts with a wall the wall may be replaced by an equal and opposite jet. Thus, Eq. (17) reduces to the following form:

$$\Delta\alpha_j = \alpha_j\left(1 - \frac{S_j}{S_j + r_{1/2}}\right)^B \tag{18}$$

In the above expression S_j is the distance at which the jth jet will interact with the wall.

III. Results and Discussion

The emphasis now is on the practical aspect of utilizing the analysis in an initial dilution zone design or exit temperature pattern development problem. In order for a designer to acquire confidence in a model and its assumptions as a predictive analysis, it is imperative to assess experimentally the range of validity of the mathematical assumptions and approximations. Unfortunately, there are essentially no experimental data that describe the trajectories of jets through the complex three-dimensional confined environment representative of a combustor dilution zone and its transition section. However, there has been limited experimentation on isolated jets within a two-dimensional confinement which can be used to assess some aspects of the analysis. Figure 6 compares such experimental trajectory data[9] with analytical predictions for three momentum ratios representative of a combustor dilution zone. The correlation between analysis and experiment is excellent even though there is some discrepancy in the near-entry region. This good agreement lends confidence to the models' capability of predicting the crossflow and wall influence on a two-dimensional jet trajectory.

Figure 7 compares the calculated trajectories for an isolated jet and opposing jets of equal mass flow and momentum flux penetrating into a circular cylinder. The isolated jet, momentum ratio 3.6, penetrates through the cylinder centerline in contrast to the 2.4 momentum ratio jet. The force of an opposing jet alters the trajectories as shown in the figure. In this case the stronger jet does not penetrate to the cylinder centerline and in fact must theoretically remain at least a half-radius from the centerline. It is noticeable that the trajectory of the higher momentum jet has been altered significantly more by the opposing jet than the lower momentum case. This is to be expected and is in agreement with the experimental results shown in Ref. 2. Figure 8 depicts the trajectories of a system of uniform, equally spaced jets penetrating into and interacting within a circular cylinder. The momentum ratio and total mass flow are constant for all cases. Since the momentum ratio is the most dominating factor affecting penetration, the trajectories for the three cases are very similar. The quantity of flow entrained for each case is a function of the number of jets as enumerated on the figure.

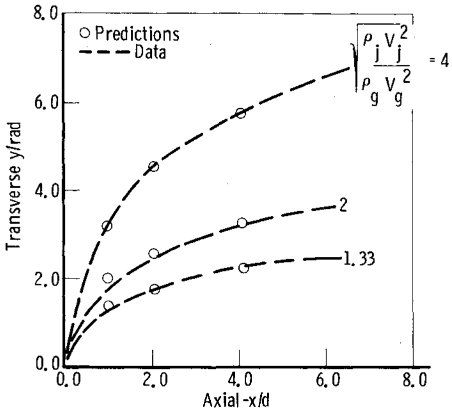

Fig. 6 Comparison of predicted and experimental jet trajectories.

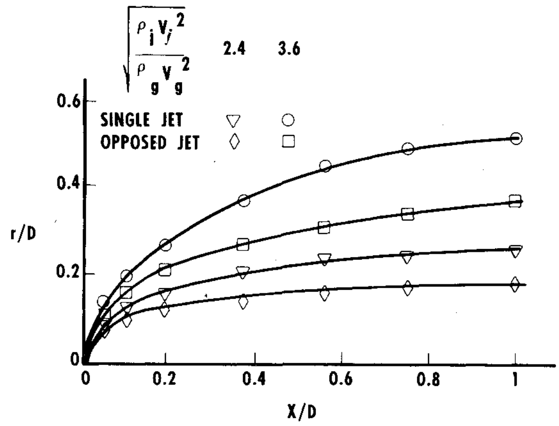

Fig. 7 Predicted trajectories for opposing jets.

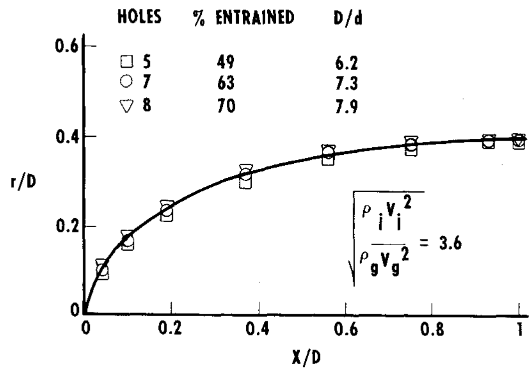

Fig. 8 Predicted trajectories for a system of jets penetrating into a circular cylinder.

A can combustor from a can-annular combustion system will be used to demonstrate the analytical capability in an actual design situation. The combustor geometry was depicted in Fig. 1. In a design application, the total effective area of the

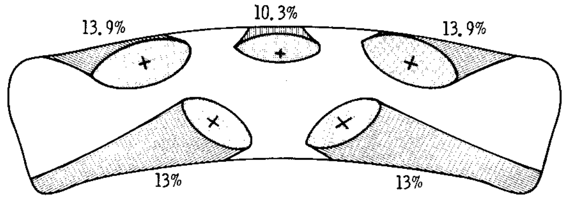

Fig. 9 Mixing coverage for a circumferentially uniform five-hole dilution zone.

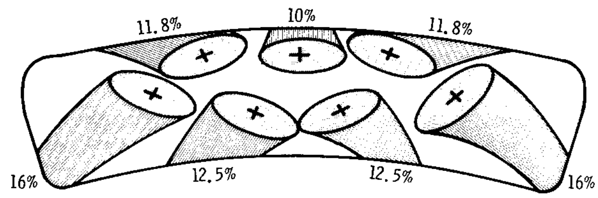

Fig. 10 Mixing coverage for a circumferentially uniform seven-hole dilution zone.

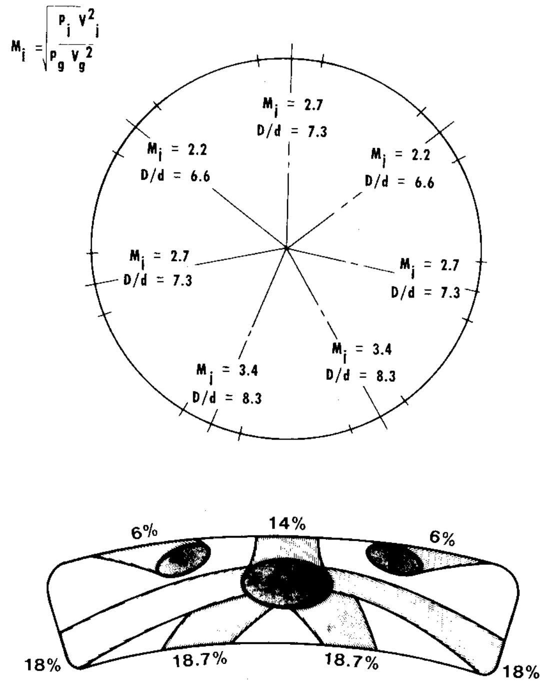

Fig. 11 Mixing coverage for a circumferentially non-uniform annulus—seven-hole dilution zone design.

dilution holes and the combustion liner pressure drop will have been defined by other combustor considerations. Also, it has been assumed here that the hot gas entering the dilution zone has uniform profiles. This is not the case, in practice, where radial and circumferential nonuniformities exist.

Figures 9 and 10 illustrate the location of the jet centerlines, percent of hot gas entrained by each jet, and approximate mixing coverage at the exit plane of the combustor transition section for a five- and seven-hole dilution zone, respectively. For both cases, the holes are uniform in size and equally spaced with one at top dead center. Also, the annulus flow is uniform. The ellipses shown are the half-radius boundaries, and their orientation in the plane is predicted by the analysis. The wake region is approximated by the shadow of the jet being convected downstream along the streamlines. Since turbulent mixing is not included, areas where there appears to be no dilution coverage should not be interpreted as having hot primary stream temperatures. In order to quantitatively evaluate the dilution zone design and predict temperature profiles, a turbulent mixing model is essential, especially at large X/D. These two figures illustrate the general analytical capability and, therefore, we can compare and contrast them as to the ability to predict the effectiveness of a design. Foremost, the five-hole dilution zone entrained 64.1% of the hot gas, while the seven-hole design entrained 90.6%. This implies that the five-hole design would provide insufficient coverage, allowing hot gas to pass through escape regions and not adequately mix prior to entering the turbine. Indiscriminately increasing the number of holes creates blockage around the perimeter and forces the hot gas through a central core at an increased momentum, thus inhibiting penetration and mixing. Since the momentum ratio is the dominating factor affecting penetration, the slight decrease in hole diameter for the seven-hole design does not appreciably influence the centerline location. However, the additional jet interactions modify the trajectories. It is anticipated that a radial profile for the five-hole design would be hub-oriented, since the majority of entrainment has occurred in the upper portion of the exit plane.

A more practical design situation for the can combustor is shown in Fig. 11. These seven unequally sized and spaced holes are designed to include the effect of annulus flow distortion which is prevalent in a can-annular combustion system. The holes are sized for equal mass flow. The distributions of hole size and momentum ratio are also shown in Fig. 11. Penetration is nonuniform since the momentum ratios vary circumferentially around the combustor. This allows jets to overlap and fall in behind one another as shown by the center core, which is expected to be a cool region of the pattern.

In order to fully utilize the analytical capability, the predicted jet coverage must be compared with a measured temperature pattern. This comparison will determine what areas need to be adjusted and if it is possible through a dilution zone modification to meet temperature pattern goals. One should recognize that the dilution zone itself cannot correct all temperature pattern problems. In some cases a modification to the primary zone is necessary. The consequence of this primary zone modification could be analyzed by modifying the initial entry conditions to the dilution zone analysis.

IV. Conclusion

An analytical technique has been developed which combustion engineers may use to investigate dilution zone designs economically. A three-dimensional interacting jet analysis predicts the trajectories of dilution jets through the dilution zone and transition section of a gas turbine combustor. The present capability to predict the jet trajectories qualitatively aids the designer in the placement of air holes when developing a combustor to meet engine durability criteria. Calibration of the model with key scientific experiments and combustor rig tests will eventually lead to a verified systematic dilution zone design technique resulting in a definite development time and cost savings.

References

[1] Norgren, C.T. and Humenik, F.M., "Dilution-Jet Mixing Study for Gas-Turbine Combustors," NASA TN-D-4695, Aug. 1968.

[2] Kamotani, Y. and Greber, I., "Experiments on Confined Turbulent Jets in Cross Flow," NASA CR-2392, March 1974.

[3] Odgers, J. and Wojcikowski, K.A., "Jet Mixing as Defined by Water Models," ASME Paper 75-WA/GT-2, presented at ASME Winter Annual Meeting, Houston, Texas, Nov. 1975.

[4] Walker, R.E. and Kors, D.L., "Multiple Jet Study Final Report," NASA CR-121217, June 1973.

[5] Adder, D. and Baron, A., "Prediction of a Three Dimensional Circular Turbulent Jet in Crossflow," *AIAA Journal,* Vol. 17, Feb. 1979, pp. 168-174.

[6]Bowley, W.W. and Sucec, J., "Trajectory and Spreading of a Turbulent Jet in the Presence of a Crossflow of Arbitrary Velocity Distribution," ASME Paper 69-GT-33, presented at the Gas Turbine Conference, Cleveland, Ohio, March 1969.

[7]Holdeman, J.D. and Walker, R.E., "Mixing of a Row of Jets with a Confined Crossflow," *AIAA Journal,* Vol. 15, Feb. 1977, pp. 243-249.

[8]Cox, G.B., Jr., "Multiple Jet Correlations for Gas Turbine Engine Combustor Design," ASME Paper 75-GT-45, presented at ASME Gas Turbine Conference, Houston, Texas, March 1979.

[9]Hunter, S.C., et al., "Advanced, Small, High-Temperature-Rise Combustor Program Vol I Analytical Model Derivation and Combustor-Element Rig Test," USAAMRDL Tech. Rept. AD778766, 1974.

[10]Wu, C.-H. and Wolfenstein, L. "Applications of Radial-Equilibrium Conditions to Axial-Flow Compressor and Turbine Design," NACA Rept. 955, Wash., D.C., 1955.

[11]Novick, A.S., Miles, G.A., and Lilley, D.G., "Numerical Simulation of Combustor Flow Fields: A Primitive Variable Design Capability," *Journal of Energy,* Vol. 3, March-April 1979, pp. 95-105.

[12]Adkins, R.C., "The Effect of Compressor Exit Flow Distribution on Air Placement in Annular Combustors," ASME Paper 78-GT-84, presented at ASME Gas Turbine Conference, London, England, April 1978.

[13]Braun, G.W. and McAllister, J.D., "Cross Wind Effects on Trajectory and Cross Sections of Turbulent Jets," NASA-SP-218, Sept. 1969, pp. 141-164.

[14]Schlichting, H., *Boundary Layer Theory,* McGraw Hill Book Co., Inc., New York, 1960.