Effects of Inlet Air Swirl and Spray Cone Angle on Combustion and Emission Performance of a Liquid Fuel Spray in a Gas Turbine Combustor

D Chatterjee, *Non-member*
A Datta, *Non-member*
A K Ghosh, *Fellow*
S K Som, *Fellow*

An extensive study is carried out to investigate the effect of inlet air swirl and fuel injection angle on the flow and combustion phenomena of a typical diffusion-controlled spray combustion process in a can-type gas turbine combustion chamber. A Eulerian-Lagrangian formulation is used for the two-phase (gas-droplet) flow, in which the coupling between the two-phases is taken care of through interactive source terms. The standard $k - \varepsilon$ model with standard wall function treatment is used to model turbulence. The initial spray parameters are specified by a suitable size distribution and a given spray cone angle. The gas phase chemical reaction is modelled using eddy dissipation model. Radiation model for the gas phase, based on the discrete ordinate method, is adopted in consideration of the gas phase as a grey absorbing emitting medium. Thermal and prompt NO_x are modelled following extended Zeldovich mechanism and Fenimore reaction.

Keywords: Spray combustion; Emission performance; Swirling flow; Gas turbine; Spray cone angle

NOTATION

C_f : fuel vapour concentration
C_{NO} : NO_x concentration
d : dia of the liquid droplet
d_{SMDi} : Sauter mean dia
D : combustor dia
P_{in} : combustor inlet pressure
r : radial coordinate
Re_{in} : inlet Reynolds number
S : Swirl number
T : mean temperature
T_a : ambient temperature
z : axial coordinate
ψ : spray cone angle

INTRODUCTION

A gas turbine combustor using liquid fuel is a complex combustion device within which there exists a wide range of interacting coupled physical and chemical phenomena like fuel spray atomization and vapourization, two-phase turbulent transport, radiation and chemical kinetics. Despite the continued advances in gas turbine combustion technology, the challenge in design remains yet open to achieve further improvement in performance in order to meet the desired requirements. Apart from the requirements of high combustion efficiency, low wall temperature and uniform exit temperature (in order to achieve a greater fuel economy), a higher propulsive power and increased combustor durability, the challenge today mainly centres around the clean emission and efficient utilization of energy resources. The emission levels of NO_x and particulate phases have to be kept minimum from the viewpoint of clean environment and overall energy economy in relation to the increasing use of low-grade fuel. Therefore, a prescription for the improvement in the design methodology calls for a physical understanding of the complex spray combustion processes in a gas turbine combustion chamber. This finally helps to identify the pertinent operating parameters and their relative influences on the important performance characteristics.

A considerable number of articles regarding the research in the area of gas turbine combustion are available in the literature, which are either experimental or numerical in nature. Experiments were conducted in typical gas turbine combustors for the study of both non-reacting (isothermal)$^{1-3}$

D Chatterjee and A K Ghosh are with the Department of Aerospace Engineering and S K Som is with the Department of Mechanical Engineering, Indian Institute of Technology, Kharagpur 721 302 and A Datta is with the Department of Power Engineering, Jadavpur University, Kolkata 700 091.

This paper was presented and discussed at the National Seminar on 'Indian Aerospace Vehicles : Technological Challenges' held concurrently with the Seventeenth National Convention of Aerospace Engineers, Ranchi, November 3-5, 2003.

and reacting flows either with gaseous or liquid fuels[4-6]. Numerical solution of the flow field in gas turbine combustors has found attention of many researches[7-8]. Their studies throw light on the flow field with typical recirculating zones in the combustor. Researchers have developed already a number of theoretically predictive models on spray evaporation and combustion. Faeth[9] suggested a comprehensive review of these models. The control of pollutants in the process of combustion of fossil fuels is one of the greatest challenges in today's technological scenario. The NO_x is identified as the major source of environmental pollution from gas turbines. The computation of thermal NO_x using Zeldovich mechanism was made by Ramos[10] and Sokolov, *et al*[11].

It appears from above discussion that, despite a large number of findings being available in the literature, studies on some specific aspects in swirl spray combustion pertaining to a gas turbine combustor require further attention.

PHYSICAL STATEMENT AND ASSUMPTIONS

The physical problem refers to the evaporation and combustion of a continuously injected liquid fuel spray in a can type combustor (Figure 1). The size and shape of the combustor is considered in conformity to the available literature[3]. The air supply to the combustor is splitted among the swirler at entry and through two radial jets in the form of secondary and dilution air. Fuel spray is injected from an atomizer located at the hub of the swirler. The fuel is considered to be n-Hexane (C_6H_{14}).

The numerical model is based on the following assumptions.

- The problem is considered to be axi-symmetric. Hence, the radial jets are taken to be uniform over the periphery.
- The fuel spray is considered to consist of a finite number of droplet classes with size distribution guided by an initial PDF described by the four-parameter Rosin-Rammler function. However, the initial velocities and temperatures of all the droplets are taken as same.
- The liquid droplets are too small in size so that the effect of gravity on them is neglected.
- Virtual mass force and Basset force on the particles are not considered due to high density ratio between the two phases.

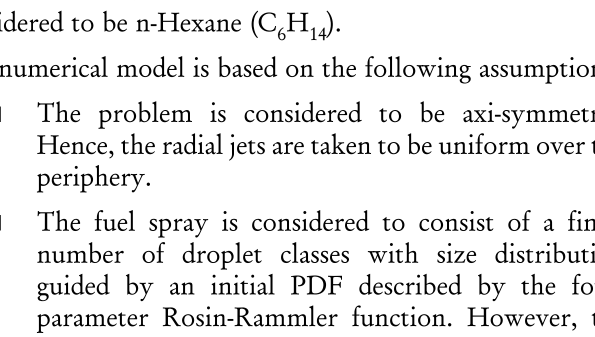

Figure 1 Schematic representation of a combustor

- Droplet collisions and coalescence are neglected, as the spray is dilute in nature.
- There is no nucleation, break-up and micro-explosion of droplets.
- Droplets do not take part in radiative energy exchange which is solved as a purely gas phase phenomenon. This is justified in a dilute spray as suggested by Faeth[9].

PHYSICAL MODEL

In this section, the basic physical model pertinent to the present problem is presented. A comprehensive detail of the governing equations of flow and combustion can be found following the guidelines suggested by Datta[12].

Swirling Flow

Swirling flows are common in combustion, with swirl introduced in burners and combustors in order to increase residence time and stabilize the flow pattern. The present analysis is modelled as a 2-D axi-symmetric problem that includes the circumferential or swirl velocity. The assumption of axi-symmetry implies that there are no circumferential gradients in the flow, but there are non-zero circumferential velocities.

Turbulence

Normally, standard $k - \varepsilon$ model is considered to be the most widely used turbulence model. In a swirling flow field with recirculation, the prediction due to standard $k - \varepsilon$ method is found to be poor in certain regions. The deficiency of standard $k - \varepsilon$ model stems from the neglect of ani-sotropic viscosity and the generation of additional turbulence due to the effect of streamline curvature. Accordingly, some advanced turbulence models can be tried out to capture the swirling recirculating flows. The appropriate choice depends on the strength of the swirl, which can be gauged by the Swirl number(s). For flows with weak to moderate swirl ($S < 0.5$), both the RNG $k - \varepsilon$ model and the realizable $k - \varepsilon$ model yield appreciable improvements over the standard $k - \varepsilon$ model. For highly swirling flows ($S > 0.5$), the Reynolds stress model (RSM) is strongly recommended[13]. The effects of strong turbulence ani-sotropy can be modelled rigorously only by the second-moment closure adopted in the RSM. In this study, the standard $k - \varepsilon$ model with standard wall function treatment has been considered. Further studies can be carried out using the aforesaid higher order turbulence models. The following model constants are used in accordance with the standard $k - \varepsilon$ model:

$$c_\mu = 0.09$$

$$c_{1\varepsilon} = 1.44$$

$$c_{2\varepsilon} = 1.92$$

$$\sigma_k = 1.00$$

$$\sigma_\varepsilon = 1.30$$

Radiation

The radiative energy exchange has been evaluated within the gas phase neglecting the influence of droplets and assuming the gas phase to be a gray, absorbing-emitting medium. The optical thickness (*aL*) is a good indicator of which radiation model to use. Here, *a* is the absorption coefficient and *L* is an appropriate length scale for the domain of the problem under consideration. For the case of flow in a combustor, it is the dia of the combustion chamber. In the present analysis, the discrete ordinate radiation model has been used, which is appropriate for optically thin ($aL < 1$) problems. The discrete ordinates (DO) radiation model solves the radiative transfer equation (RTE) for a finite number of discrete solid angles, each associated with a vector direction fixed in the global Cartesian system. The implementation in FLUENT uses a conservative variant of the discrete ordinates model called the finite-volume scheme[14,15] and its extension to unstructured meshes[16].

Discrete Phase

The fuel spray (injected continuously into the combustor) is considered to consist of a finite number of droplet classes with definite size ranges. The initial droplet size distribution of liquid fuel spray is assumed to follow a four parameter Rosin-Rammler distribution function[12]. Turbulent dispersion of particles can be modelled using either a stochastic discrete-particle approach[17] or a 'cloud' representation of a group of particles about a mean trajectory. In addition, these approaches can be combined to model a set of 'clouds' about a mean trajectory that includes the effects of turbulent fluctuations in the gas phase velocities. In this study, a stochastic discrete particle approach[13,18] is used. In this approach, the trajectory of the discrete phase particles (droplets) has been predicted by integrating the trajectory equations for individual particles using the instantaneous fluid velocity along the particle path during the integration. By computing the trajectory in this manner for a sufficient number of representative particles (termed the 'number of tries'), the random effects of turbulence on the particle dispersion may be accounted for. In the present analysis, the Discrete Random Walk (DRW) model[18] has been used. In this model, the fluctuating velocity components are discrete piece-wise constant functions of time. Their random value is kept constant over an interval of time given by the characteristic lifetime of the eddies. The DRW model may give non-physical results in strongly inhomogeneous diffusion-dominated flows, where small particles should become uniformly distributed. Instead, the DRW will show a tendency for such particles to concentrate in low-turbulence regions of the flow.

Species Transport

The species transport with the chemical reaction has been modelled using the Generalized Finite Rate Chemistry (GFRC) formulation. The finite rate chemistry approach is based on the solution of species transport equations for reactants and product concentrations. The reaction between the fuel vapour and oxidizer is considered to be a single step, irreversible, global reaction as

$$2C_6H_{14} + 19O_2 \rightarrow 12CO_2 + 14H_2O$$

The reaction rate may be controlled either by chemical kinetics or by turbulent eddies whichever is slower. In the present case, the turbulent eddy dissipation rate is much slower than the rate determined from chemical kinetics (Arrhenius rate)[12]. Hence, the eddy dissipation concept of Magnussen and Hjertager[19] is used to calculate the reaction rate. The species conservation equations are solved for fuel vapour, oxygen, carbon dioxide and water vapour, while nitrogen concentration is obtained by difference. The standard model constants[19] are used for the eddy dissipation model.

NO_x Model

Calculations are done for thermal as well as prompt NO_x formation. The formation of thermal NO_x is determined by a set of highly temperature-dependent chemical reactions, known as the extended Zeldovich mechanism[20]. The presence of a second mechanism leading to NO_x formation is first identified by Fenimore[21] and is termed prompt NO_x. The mass transport equation for the NO species, taking into account convection, diffusion, production and consumption of NO and related species is solved.

METHOD OF SOLUTION

In this section, the detailed study of numerical mesh, the numerical scheme and operating parameters along with boundary conditions are discussed.

Numerical Mesh

A numerical mesh of 100 × 40-grid nodes is used after several experiments, which shows that further refinement in either direction does not change the velocity and scalar variables at any point in the combustor by more than 2%. The 2-D quadrilateral grids are chosen to approximate the domain. The grid spacing in both axial and radial directions are changed smoothly to minimize the final accuracy of the discretization scheme due to variable grid spacing. The variations in grid spacing are made in such a way that a higher concentration of nodes occurs near the swirler and the radial holes, as also near the solid walls.

Numerical Scheme

All the computations are carried out using FLUENT 5.5 software package[13]. Segregated, implicit, steady and axi-symmetric swirl solver is used to solve the gas (or continuous)-phase conservation equations. The coupling for the two-phase (gas-droplet) flow is done by using the discrete phase modelling capabilities of FLUENT. For all the transport equations, the convective terms are discretized according to power law discretization scheme and the diffusive terms are discretized according to central differencing scheme. The pressure velocity coupling is done by SIMPLE algorithm.

Operating Parameters and Boundary Condition

The total air flow to the combustor (0.1 kg/s) is splitted between the swirler and two radial jets according to a ratio of 5:7:8 following the suggestion available in literature[12,22]. The swirler is assumed to impart a solid body type rotation to the flow of air at inlet as is done by helicoidal swirler vanes. The axial velocity of air entering through the swirler is assumed to be in a plug flow mode. The inlet swirl number is calculated, based on the axial and tangential velocity distributions through the swirler. The temperature of the air entering the combustor is considered to be 600 K. An air-fuel ratio of 60 is used for the calculation. The temperature of the fuel during injection is considered to be 300 K. Ten classes of droplets having dia from 10 μm to 130 μm and a mean drop dia of 52 μm are injected with an injection velocity of 5 m/s at different injection angles. The dispersion parameter in Rosin-Rammler distribution function is taken as 3 according to Mugele and Evans[23]. A zero axial gradient is prescribed at the outlet for all variables. Standard logarithmic law of wall is considered at the solid walls. For the discrete phase, escape condition is given at the boundaries, that is, the particle is lost from the calculation at the point where it impacts the boundary and trap condition is given at the wall, that is, non-volatile material is lost from the calculation at the point of impact with the boundary and volatile material present in the particle or droplet is released to the vapour phase at this point. For radiation calculation, the end planes are assumed to be radiatively adiabatic and Marshak boundary condition[24] is applied at the solid combustor walls. All the calculations are done considering normal atmospheric pressure as the combustor inlet pressure. The temperature is non-dimensionalized taking droplet temperature as the reference value.

RESULTS AND DISCUSSION

Figure 2 and Figure 3 show the flow fields within the combustor. At a higher Swirl number, a toroidal recirculation zone is observed at the central region in the primary zone of the combustor. However, at lower Swirl number, there exists a recirculation zone near the wall, while the central recirculation is almost absent. A wall recirculation zone appears due to the flow in the diffuser even in the case of a weak swirl, while a high radial pressure gradient, being associated with a strong swirl, nullifies the wall recirculation and results in a strong on axis recirculation.

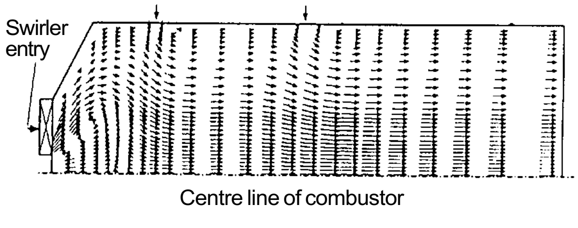

Figure 2 Flow field within the combustion chamber [$\mathrm{Re_{in}}$ = 52100, S = 0.76, d_{SMDi} = 52 μm, ψ = 80°, P_{in} = 100 kPa]

Figure 4 and Figure 5 describe the dimensionless temperature distribution within the combustion chamber for different swirl condition. At higher Swirl number, the flame spreads in the radial direction, while on the other hand, for a lower Swirl number, the flame elongates in the axial direction and, at the same time, engulfs the region near the wall in the primary zone. This can be attributed to the variation in the nature of central and wall recirculation zones with the operating parameters.

The distributions of the liner wall temperature along the length of the combustor are shown in Figure 6 and Figure 7. The wall temperature, under all situations, shows a steep rise from the inlet due to chemical reaction in the primary zone. With the introduction of secondary air, the wall temperature falls abruptly and then increases at a relatively flat rate due to the diffusion of heat from the core near the axis. A further fall in wall temperature is marked due to the introduction of dilution air followed by a flat and steady rise up to the exit. It is observed that a decrease in Swirl number or an increase in spray cone angle increases the wall temperature in the primary zone with the peak slightly shifted away from the inlet. This is because of the fact that, in case of weak swirl, the flame engulfs

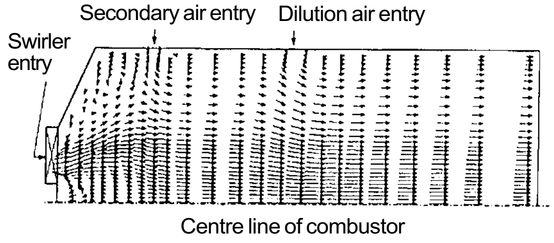

Figure 3 Flow field within the combustion chamber [$\mathrm{Re_{in}}$ = 52100, S = 0.37, d_{SMDi} = 52 μm, ψ = 80°, P_{in} = 100 kPa]

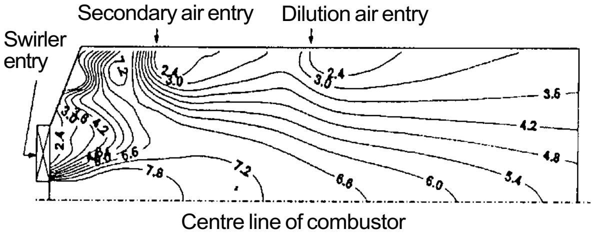

Figure 4 Contours of dimensionless temperature within the combustion chamber [$\mathrm{Re_{in}}$ = 52100, S = 0.76, d_{SMDi} = 52 μm, ψ = 80°, P_{in} = 100 kPa, T_{in} = 2 (600 K)]

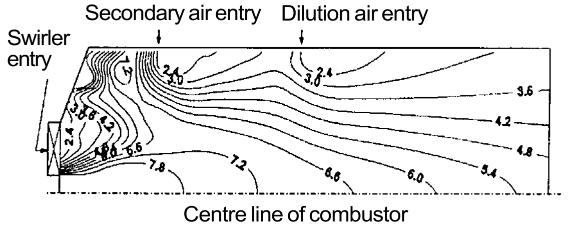

Figure 5 Contours of dimensionless temperature within the combustion chamber [$\mathrm{Re_{in}}$ = 52100, S = 0.37, d_{SMDi} = 52 μm, ψ = 80°, P_{in} = 100 kPa, T_{in} = 2 (600 K)]

the region near the wall due to the presence of a wall recirculation in the primary zone, while the flame in case of an increased spray cone angle also extends towards the wall due to large radial dispersion of fuel droplets and their subsequent vapourization close to the wall.

Figure 8 and Figure 9 show the exit temperature distribution for different swirl condition and spray cone angle. The minimum temperature occurs at the wall while maximum one occurs at the axis. This is because of the assumption of uniform blowing of secondary and dilution air in the azimuthal direction compatible to an axi-symmetric model.

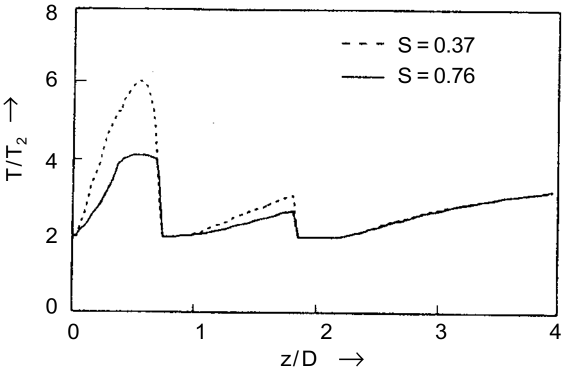

Figure 6 Variation of wall temperature along the length of combustor at different Swirl numbers

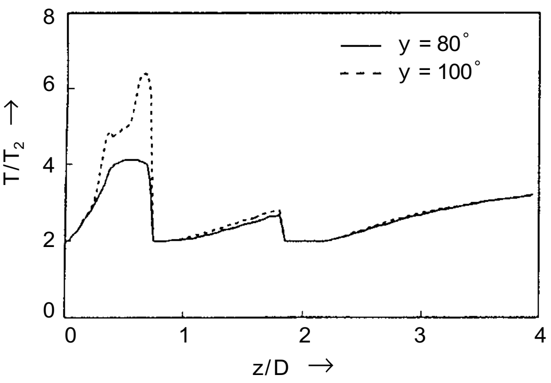

Figure 7 Variation of wall temperature along the length of combustor at different spray cone angles

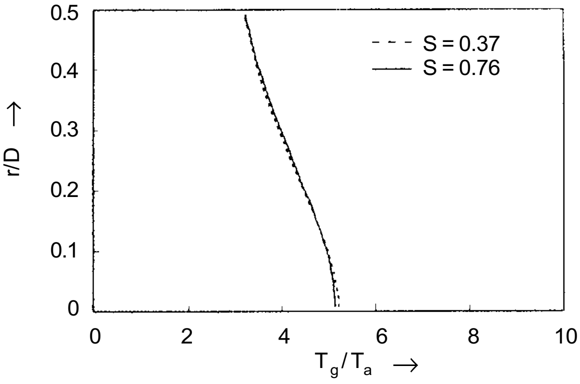

Figure 8 Exit temperature distribution for different Swirl numbers

Figure 10 and Figure 11 show the exit fuel vapour distribution for different swirl condition and spray cone angle. It has been observed that an increase in inlet air swirl increases the fuel vapour concentration at all radial locations at the exit. An increase in spray cone angle makes the profile of the fuel vapour concentration more flat with a decrease in the value of fuel vapour concentration at all radial locations, mainly near the axis. A higher spray cone angle results in a larger radial dispersion and, hence, a better mixing with the primary air. This results in the accumulation of fuel vapour towards the wall and also an enhanced rate of combustion in the primary zone. Hence, the fuel vapour concentration falls at the combustor exit.

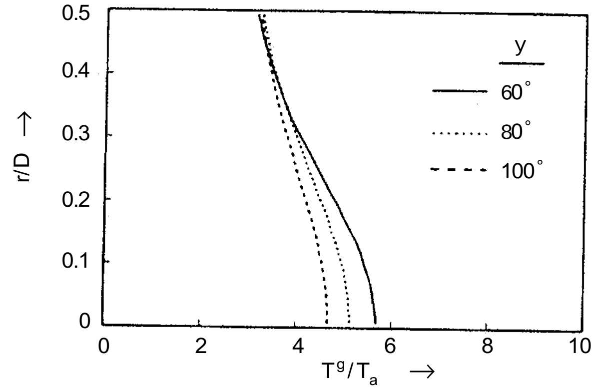

Figure 9 Exit temperature distribution for different spray cone angles

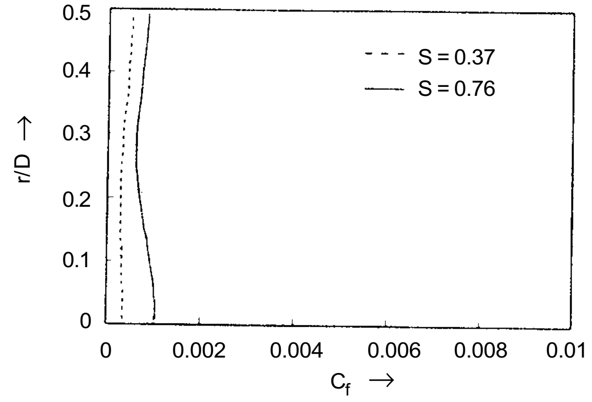

Figure 10 Exit fuel vapour distribution for different Swirl numbers

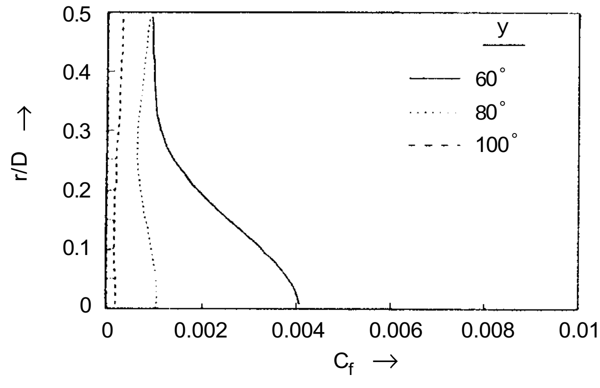

Figure 11 Exit fuel vapour distribution for different spray cone angles

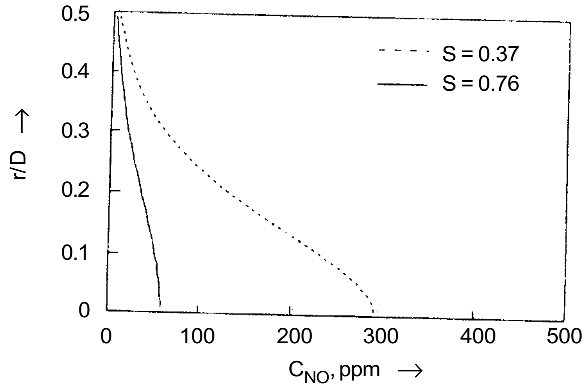

Figure 12 Exit NO distribution for different Swirl numbers

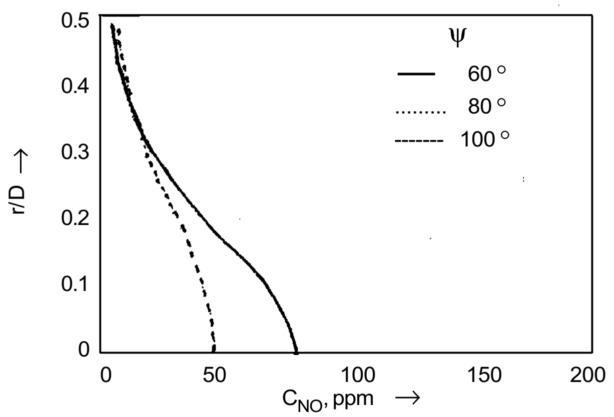

Figure 13 Exit NO distribution for different spray cone angles

Figure 12 and Figure 13 show the exit NO distribution for different swirl condition and spray cone angle. It has been observed that the exit NO concentration is hardly influenced by injection angle. An increase in the inlet air swirl increases the bulk NO concentration at the exit. Since, the formation of thermal NO is dependent mostly on the temperature, the NO concentration distribution follows the same pattern as that of the temperature.

CONCLUSION

A numerical model in simulating liquid fuel spray combustion in a tubular gas turbine combustor has been developed. The influences of inlet air swirl and spray cone angle have been depicted for the wall temperature, exit temperature, exit fuel vapour and exit NO distribution. An increase in inlet air swirl at a given combustor pressure reduces the formation of NO. However, the spary cone angle has got no influence on the exit NO concentration.

REFERENCES

1. A S Green and J H Whitelaw. 'Isothermal Models of Gas Turbine Combustors.' *Journal of Fluid Mechanics,* vol 126, 1983, p 399.

2. A F Bincen, M Senda and J H Whitelaw. 'Scalar Characteristics of Combusting Flow in a Model Annular Combustor.' *ASME Journal of Engineering for Gas Turbine and Power,* vol 111, 1989, p 90.

3. C D Cameron, J Brouwer, C P Wood and G S Samuelson. 'A Detailed Characterization of Velocity and Thermal Fields in a Model Can Combustor with Wall Jet Injection.' *ASME Journal of Engineering for Gas Turbine and Power,* vol 111, 1989, p 31

4. H Toral and J H Whitelaw. 'Velocity and Scalar Characteristics of the Isothermal and Combusting Flows in a Combustor Sector Rig.' *Combustion Science and Technology,* vol 45, 1982, p 251.

5. K H Khalil, F M El Mahallawy and H A Moneib. 'Effect of Combustion Air Swirl on the Flow Pattern in a Cylindrical Oil Fired Furnace.' '*Sixteenth International Symposium on Combustion*', *The Combustion Institute,* 1976, p 135.

6. J R Noyce, C G W Sheppard and F D Yamba. 'Measurement of Mixing and Species Concentration within a Gas Turbine Type Combustor.' *Combustion Science and Technology,* vol 25, 1981, p 209.

7. D L Rhode, D G Lilley and D K McLaughlin. 'On the Prediction of Swirling Flow-fields Found in Axisymmetric Combustor Geometries.' *ASME Journal of Fluids Engineering,* vol 104, 1982, p 378.

8. W Shyy, S M Correa and M E Braaten. 'Computations of Flow in a Gas Turbine Combustor.' *Combustion Science and Technology,* vol 58, 1988, p 97.

9. G M Faeth. 'Evaporation and Combustion of Sprays.' *Progress in Energy and Combustion Science,* vol 9, 1983, p 1.

10. J I Ramos. 'Numerical Solution of Non-premixed Reactive Flows in a Swirl Combustor Model.' *Engineering Computers,* vol 1, 1984, p 263.

11. Sokolov, *et al.* 'Mathematical Modelling of an Annular Gas Turbine Combustor.' *ASME Journal of Engineering for Gas Turbine and Power,* vol 117, 1995, p 94.

12. A Datta. 'Hydrodynamics of Swirl Spray Pressure Atomizer and Thermal-dynamics of Gas Turbine Combustion (Ph D Dissertation).' *Indian Institute of Technology,* Kharagpur, 1998.

13. 'Fluent User Manuals.' *Fluent Inc,* USA.

14. E H Chui and G D Raithby. 'Computation of Radiant Heat Transfer on a Non-orthogonal Mesh Using the Finite Volume Method.' *Numerical Heat Transfer,* vol 23, pt B, 1993, p 269.

15. G D Raithby and E H Chui. 'A Finite Volume Method for Predicting a Radiant Heat Transfer in Enclosures with Participating Media.' *Journal of Heat Transfer,* vol 112, 1990, p 415.

16. J Y Murthy and S R Mathur. 'A Finite Volume Method for Radiative Heat Transfer using Unstructured Meshes.' *AIAA-98-0860,* January 1998.

17. B J Daly and F H Harlow. 'Transport Equations in Turbulence.' *Physics of Fluids,* vol 13, 1970, p 2634.

18. Q Zhou and M A Leschziner. *Proceedings of the 8th Turbulent Shear Flows Symposium,* Munich, 1991.

19. B F Magnussen and B H Hjertager. 'On Mathematical Models of Turbulent Combustion with Special Emphasis on Soot Formation and Combustion.' *16th International Symposium on Combustion*, *The Combustion Institute,* 1976.

20. Y B Zeldovich, P Y Sandovnikov and D A Frand-Kamenetski. *Academy of Sciences of USSR,* 1947.

21. C P Fenimore. 'Formation of Nitric Oxide in Premixed Hydrocarbon Flames. *13th International Symposium on Combustion, The Combustion Institute,* 1971, p 373.

22. A Datta and S K Som. 'Effects of Spray Characteristics on Combustion Performance of a Liquid Fuel Spray in a Gas Turbine Combustor.' *International Journal of Energy Research,* vol 23, 1999, p 217.

23. R A Mugele and H D Evans. 'Droplet Size Distribution in Sprays.' *Industrial Engineering Chemistry,* vol 43, 1951, p 1317.

24. K M Case and P F Zweifel. 'Linear Transport Theory.' *Addison-Wesley Publishing Co,* 1967.