International Journal of Mechanical Engineering and Robotics Research

ISSN 2278 – 0149 www.ijmerr.com
Vol. 1, No. 1, April 2012
© 2012 IJMERR. All Rights Reserved

Research Paper

THE COMPUTATION OF SWIRLING FLOWS

Dhirgham Al-Khafaji[1]* and Abdur Rahim[2]

**Corresponding Author: Dhirgham Al-Khafaji,* ✉ *dalkhafagiy@yahoo.com*

This paper presents the application of CFD for parametric investigations in the design of combustor. A can type combustor with non-swirling and swirling flows at inlet is considered for the analysis under isothermal environment. The flow characteristic in the annulus region of combustor model is looked into. The commercially available code 'FLUENT', based on finite volume technique and incorporates the standard k- turbulence model has been used to carry out the predictions. The numerical results are validated against experimental results and a reasonable matching is observed.

Keywords: Annulus, Can-combustor, Dump diffuser, Recirculation zone, Swirl

INTRODUCTION

The role of computational methods in the design and development of gas turbine combustors has become very popular in recent years due to influences from a number of commercial Computational Fluid Dynamics (CFD) codes. From a computational viewpoint, the availability of these codes has enabled the calculation of complex flow fields with a predictive capability improved through the use of more accurate numerical schemes and physical models. Amongst the commonly used CFD software, FLUENT Inc. is used in this analysis that solves the governing conservation equations by a finite-volume formulation on a structured, non-orthogonal, curvilinear coordinate grid system using a collocated variable arrangement.

The combustor, an integral part of the gas turbine power-generating unit, does the vital task of converting the chemical energy of fuel to heat energy. It is highly desirable to achieve this efficiently to obtain better overall performance and low emission, smoke-free combustion. The combustor receives air from a compressor and delivers the burnt gases to the turbine. The rise in air temperature is achieved by burning the fuel in the liner of the

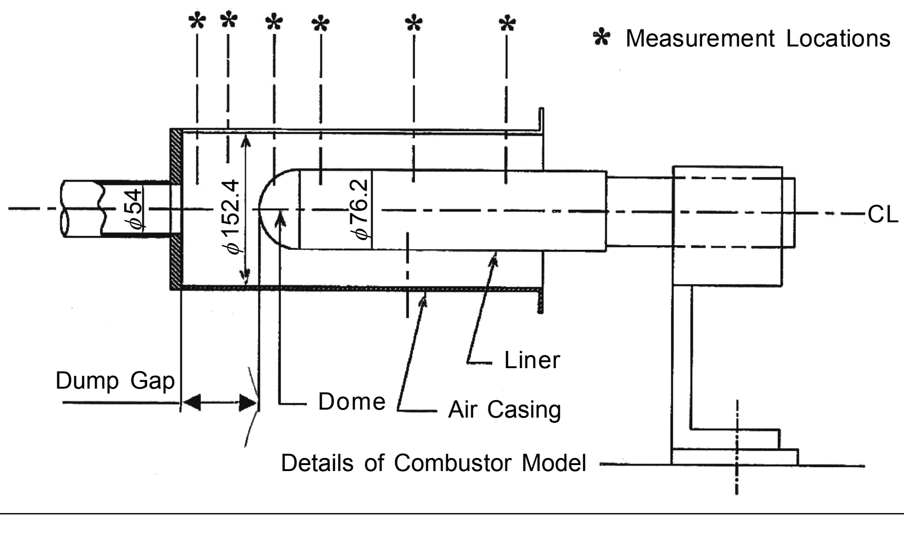
1 Department of Mechanical Engineering, University of Babylon, Bibel, Iraq.
2 Department. of Mechanical Engineering, Jamia Millia Islamia, New Delhi, India.

combustor. Initial burning occurs stoichiometrically with around one third of the compressor discharge. The combustion products are then mixed with the remaining compressed air in stages in the liner to arrive at a suitable temperature and velocity profile at the exit of the combustor for the turbine. Therefore, the primary function of a gas turbine combustor is to achieve a mixture, which can sustain continuous combustion, and to maintain, or stabilize, this combustion over a wide range of operating conditions. This is achieved by controlled mixing of fuel and air. The flow pattern in the annulus has a substantial effect on the liner flow pattern and influences the level and distribution of liner wall temperature, Lefevre (1983). From the above discussion it is quite clear that any improvement in the efficiency of the combustor requires a good understanding of the flow behavior in the annulus passage as it feeds air to the liner through primary, secondary and dilution holes. Air is fed to the liner at various stages through the annulus for complete combustion as well as for required inlet conditions at the turbine inlet. A desired condition for the annulus flow is to obtain a uniform flow in the annulus as early as possible so that flow through various liner holes can be ensured and combustor length can be reduced. Numerical simulations of combustor flow fields have been made by Novice *et al.* (1979) using $k \sim \in$ model. Prediction on recirculation zone and effects of various geometrical parameters has been observed. Karki *et al.* (1992) used $k \sim \in$ standard turbulence model for predictions of diffuser-combustor interaction with a simplified axisymmetric simulation. They found that the flow characteristics from the axisymmetric simulation are qualitatively similar.

PHYSICAL DESCRIPTION OF PROBLEM

In a can-combustor, the air leaving the compressor is split into several streams and each stream is supplied to a separate cylindrical can type combustion chamber. Figure 1 shows the can-combustor model along with dump diffuser and the critical

Figure 1: Can Combustor Model



geometrical details in which flow has been analyzed. It consists of inlet pipe of diameter 54 mm followed by a cylindrical air casing of diameter 152.4 mm and length 457.2 mm. This forms the dump diffuser of area ratio 8.16. To model the liner, a hollow pipe of 76.2 mm outer diameter and 600 mm length with hemispherical dome shape is inserted co-axially with the casing, giving rise to an annulus gap of 38.1 mm between casing and liner. This geometry is similar as the geometry of Rahim (2005) in which experiments have been carried out.

GOVERNING EQUATIONS

CFD code "FLUENT" provides comprehensive modeling capabilities for a wide range of incompressible laminar and turbulent fluid flow problems. In Fluent, a broad range of mathematical models for transport phenomena (like Heat transfer, swirl and chemical reactions) is combined with the ability to model complex geometries. The detailed description of the code is given in Fluent User's Manual (1998) and hence is only described very briefly here.

CONSERVATION OF MASS

The equation for conservation of mass, or continuity equation, can be written as follows:

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i}(\rho u_i) = S_m$$

The above equation is the general form of the mass conservation equation and is valid for incompressible as well as compressible flows. The source S_m is the mass added to the continuous phase from the dispersed second phase (e.g., due to vaporization of liquid droplets) and any user-defined sources. For 2-D axisymmetric geometries, the continuity equation is given by:

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x}(\rho u) + \frac{\partial}{\partial t}(\rho v) + \frac{\rho v}{r} = S_m$$

Here x is the axial coordinate, r is the radial coordinate, u is the axial velocity and v is the radial velocity.

CONSERVATION OF MOMENTUM

Conservation of momentum in the direction in an inertial (non accelerating) reference frame can be written as follows:

$$\frac{\partial \rho}{\partial t}(\rho u_i) + \frac{\partial}{\partial x_j}(\rho u_i u_j)$$

$$= -\frac{\partial p}{\partial x_j} + \frac{\partial \tau_{ij}}{\partial x_j} + \rho g_i + F_i$$

where p is the static pressure, τ_{ij} is the stress tensor and ρg_i and F_i are the gravitational body force and external body forces respectively. The stress tensor is given by:

$$\tau_{ij} = \left[\mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)\right] - \frac{2}{3}\mu\frac{\partial u_l}{\partial x_l}\delta_{ij}$$

where μ is the molecular viscosity.

For 2-D axisymmetric geometries the axial and radial momentum conservation equations are given by:

$$\frac{\partial}{\partial t}(\rho u) + \frac{1}{r}\frac{\partial}{\partial x}(r\rho uu) + \frac{1}{r}\frac{\partial}{\partial r}(r\rho vu)$$

$$= -\frac{\partial p}{\partial x} + \frac{1}{r}\frac{\partial}{\partial x}\left[r\mu\left(2\frac{\partial u}{\partial x} - \frac{2}{3}(\nabla \bullet \bar{v})\right)\right]$$

$$+ \frac{1}{r}\frac{\partial}{\partial r}\left[r\mu\left(\frac{\partial u}{\partial r} + \frac{\partial v}{\partial x}\right)\right] + F_x$$

$$\frac{\partial}{\partial t}(\rho v)+\frac{1}{r}\frac{\partial}{\partial x}(r\rho uv)+\frac{1}{r}\frac{\partial}{\partial r}(r\rho vv)$$

$$=-\frac{\partial p}{\partial r}+\frac{1}{r}\frac{\partial}{\partial x}\left[r\mu\left(2\frac{\partial v}{\partial r}-\frac{2}{3}\left(\nabla\bullet\bar{v}\right)\right)\right]$$

$$-2\mu\frac{v}{r^2}+\rho\frac{w^2}{r}$$

$$+\frac{2}{3}\frac{\mu}{r}\left(\nabla\bullet\bar{v}\right)+\frac{1}{r}\frac{\partial}{\partial x}\left[r\mu\left(\frac{\partial v}{\partial x}+\frac{\partial u}{\partial r}\right)\right]+F_r$$

$$\frac{\partial}{\partial t}(\rho w)+\frac{1}{r}\frac{\partial}{\partial x}(r\rho uw)+\frac{1}{r}\frac{\partial}{\partial r}(r\rho vw)$$

$$=\frac{1}{r}\frac{\partial}{\partial x}\left[r\mu\frac{\partial w}{\partial x}\right]+\frac{1}{r^2}\frac{\partial}{\partial r}\left[r^3\mu\frac{\partial}{\partial r}\left(\frac{w}{r}\right)\right]$$

$$-\rho\frac{vw}{r}$$

The present study involves axisymmetric geometries and flow involves swirl so, there is an additional momentum conservation equation for swirl velocity to be solved, which is given as:

Inlet Velocity Profile: Velocity profile well before the dump mouth has been measured with the help of a three-hole pressure probe from wall to wall at an upstream distance of 50 mm from the dump inlet plane in the inlet pipe as shown in Figures 2a and 2b. For a non-swirling flow, $S = 0$ discrepancy between the upper half axial velocity profile and lower half is in the range of 3 to 4%. This deviation is as a result of the blockage effect of the probe. It is seen that the axial velocity is flat in the middle and falls gradually towards the wall showing the growth of wall boundary layer. The tangential velocity is nearly zero as expected. For swirling flow, $S = 0.38$, it is seen that the axial velocity on the centerline drops significantly and the peak velocity is forced outwards due to the tangential momentum of flow. The tangential velocity distribution shows a forced vortex behavior from the axis up to approximately $r/r_i = 0.55$ and then drops down to zero towards the wall due to the no-slip boundary condition.

Figure 2: Inlet Axial and Tangential Velocity Profiles for Non-Swirling and Swirling Flows

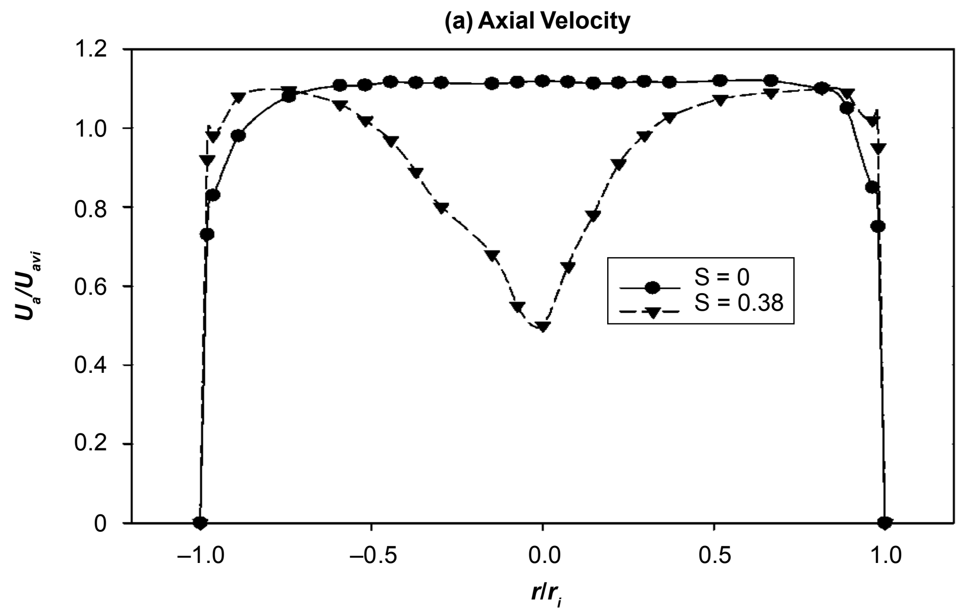
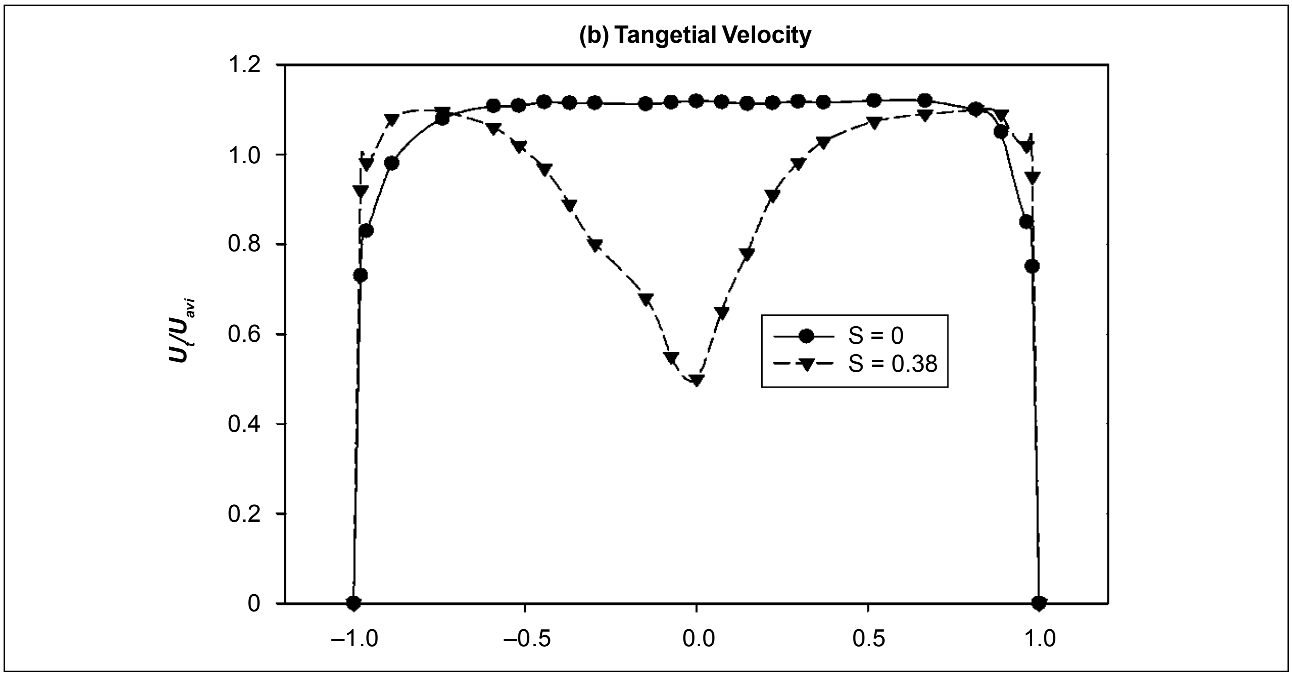


Figure 2 (Cont.)

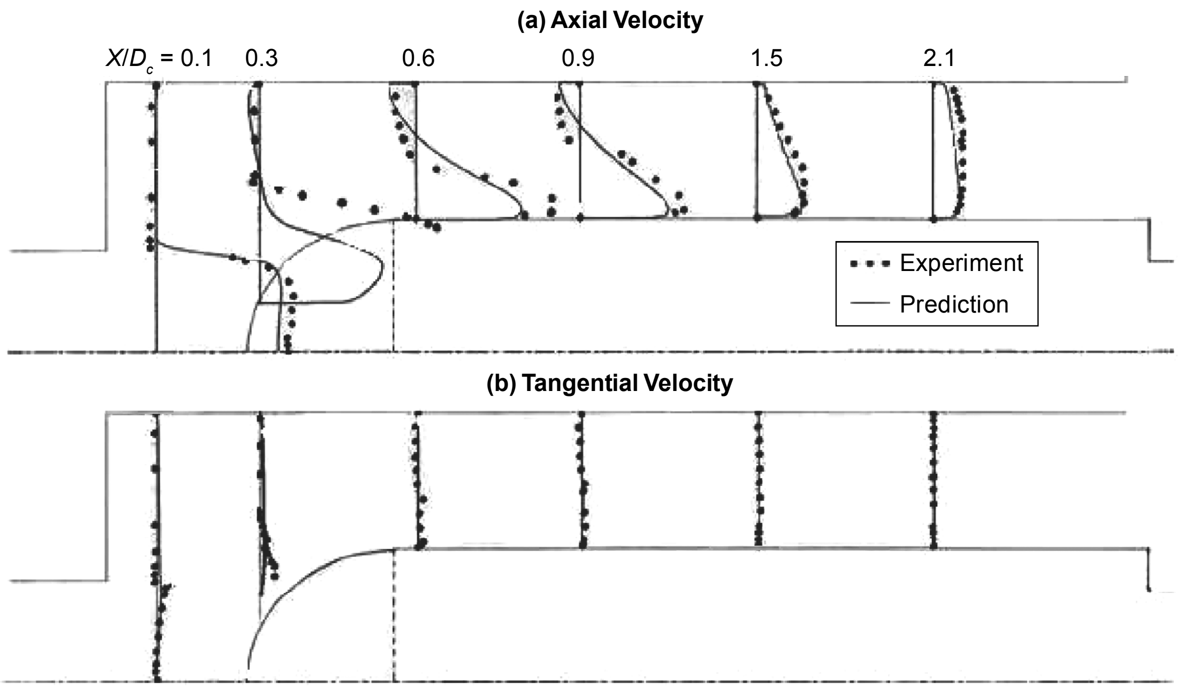


RESULTS AND DISCUSSION

Flow symmetry was established by measuring the flow from wall to wall. Here the results are presented in one half of the combustor model only. The coordinate system is as shown in Figure 1 where the axial distance is normalized with the diameter of the casing as the origin (X/D_c = 0) is located at the dump mouth and radial distance is normalized with the inlet pipe radius. Moreover, velocities are normalized with the mass—a A commercial CFD code 'FLUENT' has been used which is the state of art software package for analyzing fluid flow and heat transfer problems involving complex geometries. The numerical scheme employed belongs to a finite volume group and adopts integral from of the conservation equations. The solution domain is subdivided into a finite number of contiguous control volumes and conservation equations are applied to each control volume. Surface and volume integrals are approximated using suitable quadrature formulae. The geometry taken for the CFD analysis configures a can-combustor model as shown in Figure 1. The problem has been modeled and solved for various meshing in order to check for mesh in-dependency. Starting with course meshing, and reaching to the final finer meshing, the change in results was observed to be insignificant. The velocity at the inlet and pressure at the outlet were specified as the boundary conditions for the geometry considered. The inlet axial and tangential velocity profiles are shown in Figure 2 whereas the pressure at outlet is taken to be atmospheric. It is well known that all the turbulence models currently available have their own credibility and limitations. Although very advanced models are available for closure of the system of equations, no model is universal to be used for the flow prediction in all sorts of flow systems. Hence, it is important to identify/validate the proper turbulence model for the application of the flow prediction in the geometry being considered. Validation fundamentally means demonstration of computational fidelity to reality. In other way, code validation relies on comparison of computational results to experimental data. The validation becomes increasingly more complex due to the availability of a large number of turbulence models and solving techniques. The methodology adopted for the present investigation involves comparing the predicted performance parameters and the distribution of flow parameters at various sections of the combustor model with experimental results of Rahim (2005). The standard k-ε turbulence model is a two-equation model and it is widely used for fluid flow analysis through complex geometries. Validation for non-swirling flow in the actual can combustor model with hemispherical liner dome has been carried out with different turbulence models and it was found out that standard k-e model gives the best comparison. Figure 3 shows the comparison of experimental and numerical results of axial velocity and tangential velocity profiles for non-swirling flow for hemispherical dome liner with a dump-gap of 0.5. It is observed that comparison is good for the first measurement location (X/D_c = 0.1), which lies in the dump region. For the next measurement location (X/D_c = 0.3), which is on the dome surface, comparison is very poor may be for the following reasons:

- Presence of the steep velocity gradients due to acceleration of the flow over the dome.
- The experimental results in this region are also subject to large error as measurements have been taken by three-hole probe.

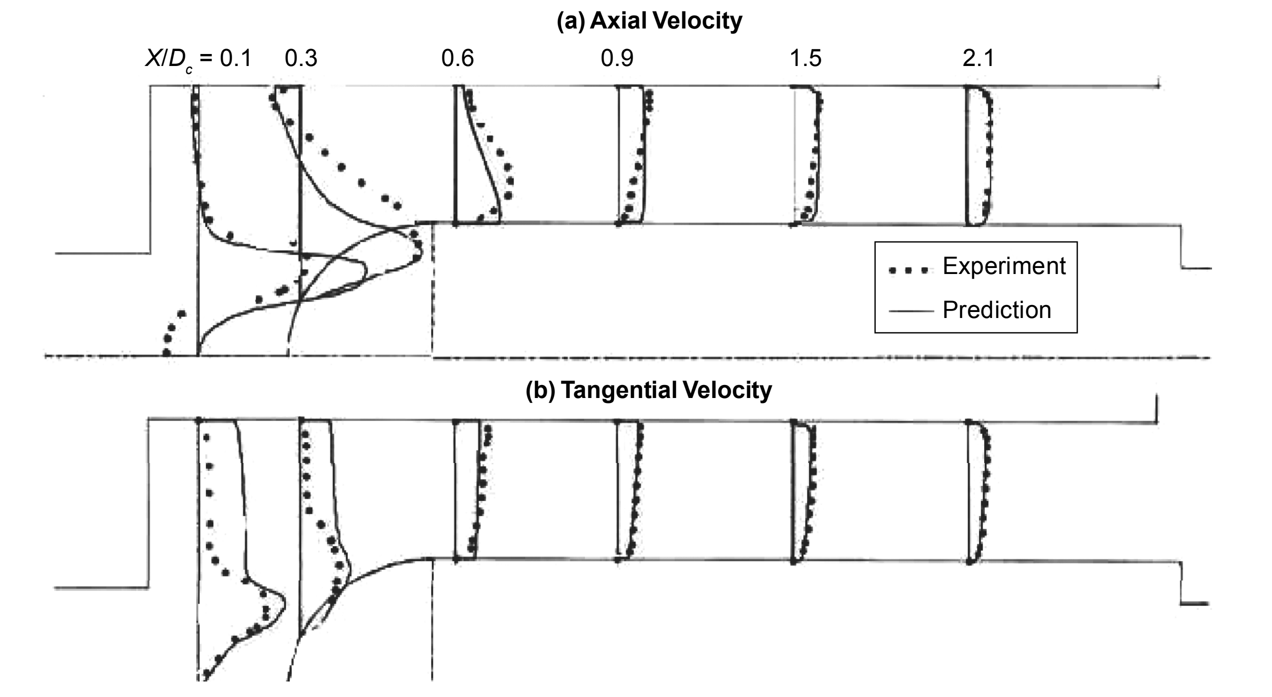
Figure 3: Comparison of Experimental and Computational Results for Non-Swirling Flow, S = 0



- Error introduced in three-hole probe measurements due to large radial velocity component.

The comparison for the downstream locations improves considerably and it is reasonably very good for the last two measuring locations. The improvement in the comparison shows that the presence of strong radial velocity component does affect the three-hole probe readings. In the annulus region, the radial component of velocity is nearly zero and hence one observes reasonable improvement in matching with experimental results. Figure 4 shows the comparison of experimental and predicted results for the swirling flow. It is observed that central recirculation zone is not picked up by the predicted result in axial velocity profile whereas corner recirculation zone is picked up qualitatively. In tangential velocity variation again the corner recirculation region is not matching quantitatively. This is due to limitations of k-ε turbulence model in recirculation zone. The velocity profile in the annular region is matching qualitatively as well quantitatively for both axial velocity and tangential velocity.

Figure 4: Comparison of Experimental and Computational Results for Swirling Flow, S = 0.38



CONCLUSION

- For non-swirling flow, a good match was obtained for the mean velocity profiles between the experimental data and numerical simulation over most of the flow (except for the high shear and high acceleration zones of the dump region).
- For a swirling flow, the matching is not very good especially in dump region and

high shear flow region where corner and central recirculation zones are observed.

- The standard k- turbulence model has been used to carry out the predictions was a good tool to simulate the swirl flow.

REFERENCES

1. Bonaldo A and Kelman J B (2008), “Experimental Characterisation of Swirl Stabilized Annular Stratified Flames”, in Press Comb, Flame.
2. Fluent Inc. (1998), “Fluent User’s Guide”, pp. 1-4, Fluent Incorporated, Lebanon NH 03766.
3. Karki K C, Oechsle V L and Mongia H C (1992), “A Computational Procedure for Diffuser Combustor Flow Interaction Analysis”, *J. Engg. for Gas Turbines and Power*, Vol. 114, pp. 1-7.
4. Lefebvre A H (1983), “Gas Turbine Combustion”, Hemisphere Publishing Corporation, Washington.
5. Novice A S, Miles G A and Lilley D G (1979), “Numerical Simulation of Combustor Flowfields: A Primitive Variable Design Capability”, *J. Energy*, Vol. 3, No. 2, pp. 95-105.
6. Rahim A, Singh S N and Veeravalli S V (2007), “Liner Dome Shape Effect on the Annulus Flow Characteristics with and Without Swirl for a Can-Combustor Model”, Proceedings of the Institution of Mechanical Engineers, *Part A: Journal of Power and Energy*, Vol. 221, pp. 359-368.
7. Roux S, Lartigue G, Poinsot T, Meier U and Berat C (2010), “Studies of Mean and Unsteady Flow in a Swirled Combustor Using Experiments, Acoustic Analysis and Large Eddy Simulations”, available at http://www.cerfacs.fr/