A Multiple Inlet Swirler for Gas Turbine Combustors

Yehia A. Eldrainy, Hossam S. Aly, Khalid M. Saqr, and Mohammad Nazri Mohd Jaafar

Abstract— The central recirculation zone (CRZ) in a swirl stabilized gas turbine combustor has a dominant effect on the fuel air mixing process and flame stability. Most of state of the art swirlers share one disadvantage; the fixed swirl number for the same swirler configuration. Thus, in a mathematical sense, Reynolds number becomes the sole parameter for controlling the flow characteristics inside the combustor. As a result, at low load operation, the generated swirl is more likely to become feeble affecting the flame stabilization and mixing process. This paper introduces a new swirler concept which overcomes the mentioned weakness of the modern configurations. The new swirler introduces air tangentially and axially to the combustor through tangential vanes and an axial vanes respectively. Therefore, it provides different swirl numbers for the same configuration by regulating the ratio between the axial and tangential flow momenta. The swirler aerodynamic performance was investigated using four CFD simulations in order to demonstrate the impact of tangential to axial flow rate ratio on the CRZ. It was found that the length of the CRZ is directly proportional to the tangential to axial air flow rate ratio.

Keywords—Swirler, Gas turbine, CFD, Numerical simulation, Recirculation zone, Swirl number

I. INTRODUCTION

CONTINUOUS combustion processes, such as those occurring in a gas turbine combustor, require the flame to be anchored at a zone within the flow field. In order to stabilize the flame, the incoming high-speed air must be decelerated to a velocity below the turbulent flame speed. The flame stabilizes along the locus of points where the air velocity is equal to the flame speed [1], [2].

This flame stabilization can be achieved by various methods. The most common techniques used in a gas turbine combustor involve creating a stagnation point. By far, the most common method to stabilize the flame in modern gas turbine combustors is swirl stabilization in which a swirl velocity is imparted to the inlet air using vane swirlers.

The degree of swirl in the flow is quantified by the dimensionless parameter, S_N, known as the swirl number which is defined as:

Yehia A. Eldrainy, Hossam S. Aly and Mohammad Nazri Mohd Jaafar are with the Department of Aeronautical Engineering, Faculty of Mechanical Engineering, Universiti Teknologi Malaysia (Corresponding Author: phone: +607 553-5843; fax: +607 553-5997; corresponding author e-mail: yeldrainy@yahoo.com, co-authors emails: hossam.saed@gmail.com, nazri@fkm.utm.my).

Khalid M. Saqr is with Department of Thermofluids, Faculty of Mechanical Engineering, Universiti Teknologi Malaysia (e-mail: mmdmskhalid2@siswa.utm.my).

$$S_n = \frac{\int_0^R \rho UW r^2 \, dr}{R \int_0^R \rho U^2 r \, dr} \quad (1)$$

where U and W are the axial and tangential velocity components respectively and R is the swirler radius. This number is a ratio of the axial flux of angular momentum to the axial thrust.

Recirculation is generally not observed for swirl numbers below 0.4, so most swirl-stabilized burners are designed for swirl numbers greater than 0.6 [1], [2]. The swirling flow spreads as it moves downstream and centrifugal force creates a low pressure zone in the centre of the flow. At a certain point downstream, the low pressure region in the centre of the flow causes the vortex to collapse inwards on itself in a process known as vortex breakdown [3]. This creates a recirculation zone in the centre of the flow. This is essential to provide sufficient time, temperature and turbulence for a complete combustion of the fuel [4], [5].

Swirl does not only help to stabilize the flame but also to produce other effects which are beneficial to the combustion system. These effects primarily include promoting fuel and air mixing and assisting the control of combustion temperatures and emissions. This is because of the strong shear regions, high turbulence and rapid mixing rates produced by the swirling vortices and the resulting toroidal recirculation zone. The various characteristics of swirl combustion are discussed extensively in the literature [6], [7].

Swirling flows result from the application of a spiral motion, a swirl velocity component being imparted to the flow by the use of some kinds of swirl generators, such as swirl vanes, axial-plus-tangential entry swirl generator or by direct tangential entry into the chamber. In gas turbine engines, vanes swirlers are the most common types of swirl generators used and can be of various designs; axial or radial; single or double (co-rotating or counter rotating). The swirl vanes are often flat for ease of manufacturing, but curved vanes may sometimes be preferred due to their potentially better aerodynamics properties.

The current vane swirlers share one disadvantage though, which is the constant swirl number independent of the amount of inlet flow rate. At relatively low loads, the inlet air flow rate is correspondingly low. This means that the produced swirl and turbulence become low, reducing the recirculation zone of intensity. For this reason, the performance of gas turbine combustors is reduced at low operating conditions (i.e. low Reynolds number).

The novel vaned swirler presented in this study divides the inlet air mass flow into two streams. One stream is introduced

conventionally to the combustor through axial vaned port. The second stream is introduced through multiple tangentially vaned ports. The tangential air flow stream intercept with the axially swirling air flow stream before entering the combustor. The fundamental major advantage in this new concept is the possibility to vary the swirl number for the same inlet mass flow using a fixed swirler configuration. When the swirl velocity component is feeble at low flow velocities, it is now possible to amplify it through the tangential air stream which will boost the value of the swirl velocity component.

II. CONFIGURATION AND GEOMETRY OF THE NEW CONCEPT

The new swirler is introduced in Figure 1. The inlet air flow is divided into two streams. One stream passes axially through a cylindrical passage with conventional vanes. The second stream passes tangential to the axial flow passage through multiple tangential ports. The axial flow stream is interrupted by the tangential flow creating a swirling turbulent region. The axial and tangential flow stream directions are illustrated in Figures 2 and 3 respectively. The intensity of swirling flow is mainly governed by the ratio of the tangential to axial air mass flow rates. Thus, it is possible to vary the swirl number for the same configuration, which is not possible in the state of the art swirlers.

Fig.1 The concept of the new swirler with a cutout in the outer shroud showing two of the tangential vanes
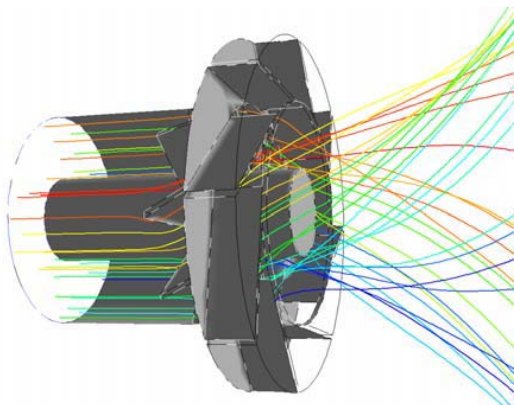

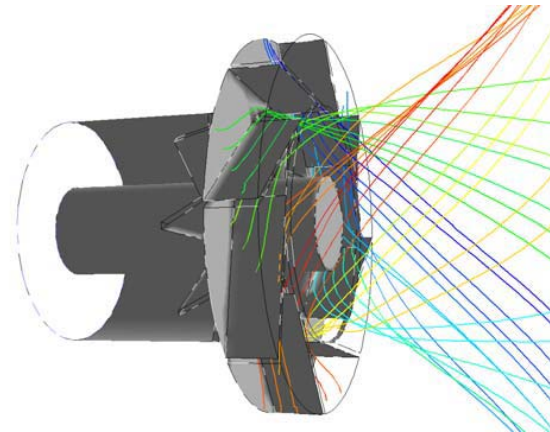Fig. 2 Axial flow pathlines passing through the axial port and vanes

Fig. 3 Tangential flow pathlines passing through the tangential

III. CFD SIMULATION

A. Physical domain and grid

The physical domain represents the flow inside the novel swirler concept, discharging to a combustor via one port. The mixing of the tangential and axial flows takes place upstream the combustor inlet port. The dimensions of the physical domain are illustrated in Figure 4. The total volume of the problem physical domain was 1.313729×10^{-2} m^3. A total number of non uniform hexahedral cells of 1040674 were implemented to get a minimum cell size of 3.924182×10^{-10} m3 in the zone of high gradients (Figure 5).
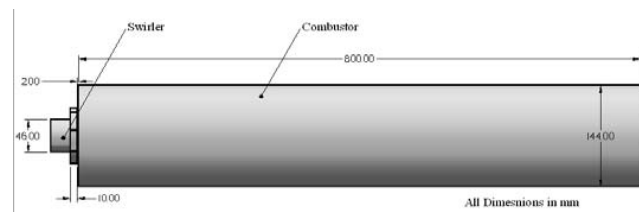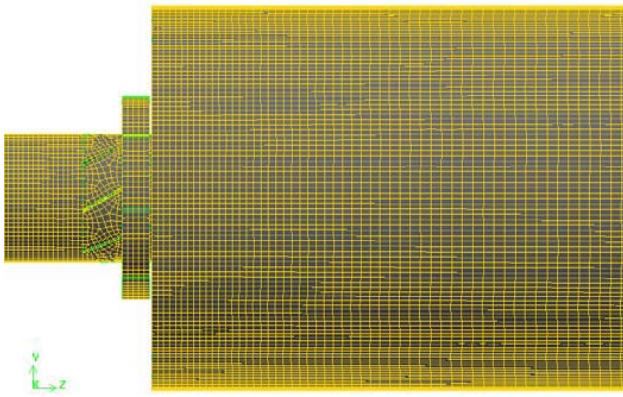

Fig. 4 Dimensions of the physical domain

(a)

(b)

Fig. 5 Computational gird

B. Governing equations

The 3D flow was assumed to be incompressible and turbulent. The governing equations are the conservation of mass and momentum as in equations (2) and (3). The turbulence model used in the present study is the standard k-ε model [8], [9]. The equations of the turbulence kinetic energy (k) and its dissipation rate (ε) are equations (4) and (5), respectively.

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i}(\rho \bar{u}_i) = 0 \tag{2}$$

$$\frac{\partial}{\partial t}(\rho \bar{u}_i) + \frac{\partial}{\partial x_i}(\rho \bar{u}_i \bar{u}_j) = -\frac{\partial p}{\partial x_j} + \tag{3}$$

$$\frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} - \frac{2}{3}\delta_{ij}\frac{\partial \bar{u}_j}{\partial x_i}\right)\right] + \frac{\partial}{\partial x_j}\left(-\overline{\rho u'_i u'_j}\right)$$

$$\frac{\partial k}{\partial t} + \bar{u}_i \frac{\partial k}{\partial x_i} = v_t \left[\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i}\right]\frac{\partial \bar{u}_i}{\partial x_j} \tag{4}$$

$$+ \frac{\partial}{\partial x_i}\left(\frac{v_t}{\sigma_k}\frac{\partial k}{\partial x_i}\right) - \varepsilon$$

$$\frac{\partial \varepsilon}{\partial t} + \bar{u}_i \frac{\partial \varepsilon}{\partial x_i} = C_{\varepsilon 1}\frac{\varepsilon}{k}v_t\left[\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i}\right]\frac{\partial \bar{u}_i}{\partial x_j} \tag{5}$$

$$+ \frac{\partial}{\partial x_i}\left(\frac{v_t}{\sigma_\varepsilon}\frac{\partial \varepsilon}{\partial x_i}\right) - C_{\varepsilon 2}\frac{\varepsilon^2}{k}$$

C. Numerical method and solver features

The CFD code Fluent 6.3 [10] was used in order to solve the discretized governing equations. Four numerical simulations were performed at four different tangential to axial mass flow ratios in order to examine the impact of the mass flow ratio on the length of the CRZ. In all the simulations, a steady state pressure based solver was used to solve the governing equations, and the Semi Implicit Method for Pressure Linked Equations (SIMPLE) algorithm [11] was used for pressure/velocity coupling. At the inlet of the computational region, the inlet boundary condition is defined

as mass flow inlet while the exit boundary is defined as pressure outlet (gauge pressure at model outlet is 0.0). Some assumptions about boundary conditions that were not directly measured had to be made as follows:

Velocity components and turbulence quantities at the inlet were uniform.

Turbulence at inlet is calculated from the following equations [12]:

$$k_{inlet} = 0.002(u^2)_{inlet} \tag{6}$$

$$\varepsilon = \frac{k_{inlet}^{1.5}}{0.3 D} \tag{7}$$

IV. RESULTS AND DISCUSSIONS

The vortex breakdown phenomenon in swirling flow is closely related to the inlet flow Reynolds number ReD and swirl number SN. Moreover, it is affected by the downstream boundary conditions [13]. To explore the effects of the new swirler design on the swirling flow field, the influence of tangential to axial inlet mass flow ratio on swirling flow has to be investigated. The next section presents a discussion of the shape of the CRZ and the formation of additional recirculation zones at the chamber corners.

The swirl number was numerically computed for the four cases using the integration shown in Equation (1) on the swirler exit plane. The different values of swirl number are listed in Table 1. It is clear that the swirl number increases with the increase of tangential air flow rate. In the present concept, the swirl number increased approximately 3.9 times when the ratio of the tangential to axial flow rates increased from 0.333 to 1.667.

The contours of reversed flow region (negative axial velocity) on stream-wise plane are shown in Figure 6 (a-d). The values of tangential and axial flow rates for the four cases are shown in Table 1.

TABLE 1
TANGENTIAL AND AXIAL FLOW RATES FOR THE FOUR CASES

case	a	b	c	d
Tangential mass flow rate	0.005	0.0075	0.01	0.0125
Axial mass flow rate	0.015	0.0125	0.01	0.0075
Tangential to axial flow ratio	0.333	0.600	1.000	1.667
Swirl number	0.4744	0.6325	0.9417	1.851

Careful assessment of Figure 6 (a-d) reveals the dependency of the CRZ length and shape on the tangential to axial mass flow rate ratio. It is evident that the CRZ is controlled in this new swirler concept by varying the inlet mass flow ratio. Thus, the CRZ and consequently the flame stability are controlled by varying the tangential to axial flow rate ratio.

Figure 6 also points out another major effect for varying the inlet mass flow ratio, namely, the diminishing of the corner recirculation zones (i.e. dead zones) as the tangential to axial

flow rate ratio increases.

(a)   (b)
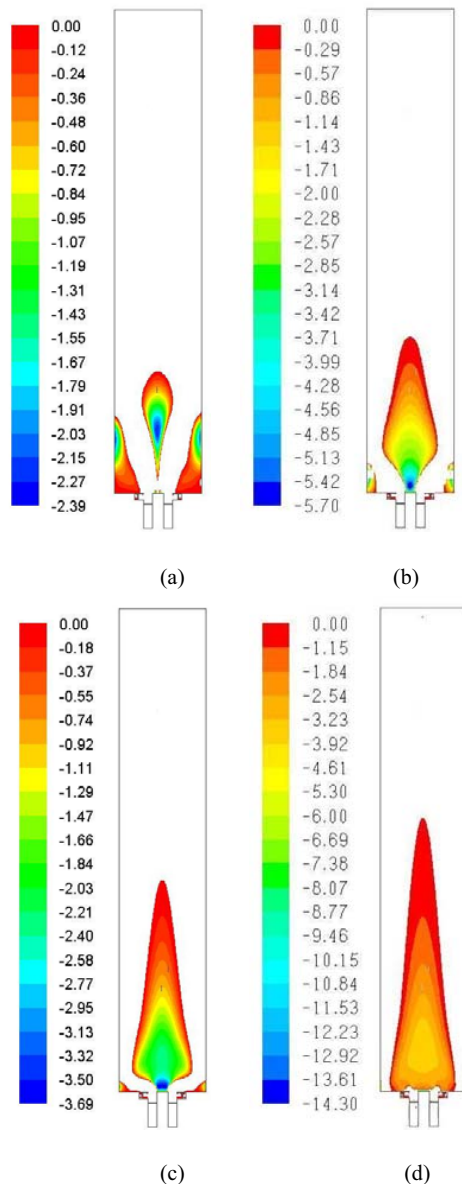
(c)   (d)

Fig. 6 CRZ for different axial to tangential air velocity ratios

These dead zones have a negative effect on the combustion process and the homogeneity of the wall temperature as well [2]. Therefore, eliminating or at least reducing these unfavorable recirculation zones is vital for enhancing the combustion process. It is noteworthy here to state that these positive effects of increasing the tangential to axial flow rate ratio come with the expense of increasing the pressure drop through the combustor, and hence reducing its efficiency. Consequently, a careful optimization study must be performed in order to decide the range of inlet mass flow ratios operational conditions.

V. CONCLUSION

A new multiple inlet swirler for gas turbine combustors was presented. The new concept enables the variation of swirl number at the same air inlet mass flow rate. The performance and main characteristics of the new swirler was examined through four numerical simulations. These simulations evidently proved that the swirler number significantly changes with the variation of tangential to axial flow rate ratio, enabling the tuning of swirl number according to turbine load. The new concept is deemed to enhance the combustion efficiency because of its ability to produce high swirl number especially at low turbine loads. In addition, the new swirler eliminates the dead zones proved to have negative effect on the combustion process. Immediate future work should comprehensively investigate the swirler isothermal performance including turbulence production and pressure loss. It is also vital to study the effect of tangential to axial mass flow ratio on the combustion efficiency, emissions and flame stability.

REFERENCES

[1]   A. H. Lefebvre, Gas turbine combustion. Hemisphere Publishing Corporation, first edition, 1983.
[2]   M. Mellor, Design of Modern gas Turbine Combustors, Academic Press, 1990.
[3]   Y. Wang, V. Yang, and R.A. Yetter, Numerical Study on Swirling Flow in an Cylindrical Chamber, 42nd AIAA Aerospace Sciences Meeting, Reno, Nevada, 2004.
[4]   Beer, J.M. and Chigier, N.A. (1972). Combustion Aerodynamics. Applied Science Publisher, London
[5]   Syred, N., Beer, J.M. (1974). Combustion in Swirling Flows: A Review. Combustion and Flame, 23, pp. 143-201
[6]   Gupta, A.K., Lilley, D.G. and Syred, N. (1984). Swirl Flows. Abacus Press, Tunbridge Wells, England.
[7]   Sloan, D.G., Smith, P.J. and Smoot, L.D. (1986). Modelling of Swirl in Turbulent Flow System. Prog. Energy Combust. Sci, Vol 12, pp. 163-250.
[8]   B. E. Launder and D. B. Spalding. Lectures in Mathematical Models of Turbulence. Academic Press, London, England, 1972.
[9]   B. E. Launder and D. B. Spalding. The Numerical Computation of Turbulent Flows. Computer Methods in Applied Mechanics and Engineering, 3:269-289, 1974.
[10] FLUENT 6.3 User's Guide, Fluent Inc. 2006.
[11] Dynamics Jiyuan, T., Guan, H. Y., Chaoqun, L., 2008 Computational Fluid: A Practical Approach, Butterworth-Heinemann Piblishing, pp 163-175.
[12] Versteeg, H.K. and Malalasekera, W., 1995, An Introduction to Computational Fluid Dynamics, the Finite Volume Method", Longman Group Ltd.
[13] Lucca-Negro and O'Doherty, 2001 O. Lucca-Negro and T. O'Doherty, Vortex breakdown: a review, Progress in Energy and Combustion Science 27 (2001), pp. 431–481.