18èmes Journées Internationales de Thermique (JITH 2017)
Monastir (Tunisie), 25 - 27 Octobre 2017

Impact of the Bluff-Body Material on the Flame Structure and Flame–Flow Interaction: Case of Methane/Air Flames

ALLICHE Mounir*, REBHI Redha

Laboratoire de Mécanique Physique et Modélisation Mathématique, université de MEDEA, Alegria
*Email: **alliche.mounir@univ-medea.dz**

Abstract:

In this work, we study the interaction between the flame structure, the flow field and the coupled heat transfer with the flame holder of a CH4 /air flame stabilized on a heat conducting bluff body in a channel. The study is conducted with a 2D CFD numerical simulation with detailed chemistry and species transport and with no artificial flame anchoring boundary conditions. Capturing the multiple time scales, length scales and flame-wall thermal interaction was done using a low Mach number assumption, coupled with a block-structured adaptive mesh refinement and an immersed boundary method for the solid body. The flame structure displays profiles of the main species and atomic ratios similar to previously published experimental measurements on an annular bluff body configuration for both laminar and turbulent flow, demonstrating generality of the resolved flame leading edge structure for flames that stabilize on a sudden expansion. The flame structure near the bluff body and further downstream shows dependence on the thermal properties of the bluff body. We analyze the influence of flow strain and heat losses on the flame, and show that the flame stretch increases sharply at the flame leading edge, and this high stretch rate, together with heat losses, dictate the flame anchoring location. By analyzing the impact of the flame on the flow field, we reveal that the strong dependence of vorticity dilatation on the flame location leads to high impact of the flame anchoring location on the flow and flame stretch downstream. This study shows the impact of heat losses to the flame holder on the flame–flow feedback mechanism in lean combustion.

Keys words: CFD, Turbulent Flame, Heat Loss, Bluff Body.

1. Introduction

Turbulent combustion in engineering applications continues to pose a very important challenge to modellers due to the multitude and complexity of processes which interact within such systems. Turbulence, chemical kinetics, thermal radiation and pollutants formation coupled with complex geometry and boundary conditions are typical of such complications. Standard approaches, based on Reynolds or Favre averaging, using first- or second-order closure models have been used extensively to compute flows in many such applications, including reactive systems [1]. These methods have manageable computational requirements, and can handle thermal radiation and complex geometry. One main disadvantage is that high temperature chemistry is very difficult to account for, as the representative source terms are highly nonlinear and hence impossible to close. Various approaches have been developed to account for turbulent combustion and the associated heat release and density fluctuations. Basic concepts, such as the flame sheet approximation, full or partial equilibrium and laminar flamelet burning [2], have been exploited to represent the behaviour of reaction zones in turbulent flame environments.

Recent gas turbine combustors use a combination of the two for a more efficient combustion process. They incorporate flame holders in the form of bluff bodies or swirlers for a primary or secondary combustion in high speed reacting flows. These two methods are very prominent among combustion systems over a long time. Researchers have been studying the effects of fluid flow inside a combustion chamber which is characterized by high turbulence caused by the wake formation. The bluff-body burner geometry is a suitable compromise as a model problem, where both methods can be tested and validated. Bluff-body flames have complex recirculating flows, similar to those found in practical combustors, but with much simpler and well defined boundary and initial conditions. Flames with significant chemical kinetic effects may be stabilized and studied without the complications of soot formation or thermal radiation. Some experimental data already exist for bluff-body flames for a range of fuels and over a range of turbulent mixing rates [3].

In bluff body combustion, a non-streamlined body is placed in a high-speed flow which produces a recirculating zone in the wake region that allows the combustion products to settle in this region for continuously igniting the incoming flow of fuel and air from which the flame propagates into the free stream. Although a premixed combustion regime is becoming increasingly popular for industrial gas turbines because of their potential for very low NOx emissions, combustion in conventional aircraft gas turbines occurs predominantly in the non-premixed diffusion flame regime.

In such devices, turbulent mixing exerts a dominant influence and better understanding necessarily requires an accurate prediction of the turbulent velocity field. Turbulent reacting flows form an important class of industrially relevant systems that are amenable to numerical simulations. With increased importance of pollutant control and process optimization, Computational Fluid Dynamics (CFD) can play a vital role in the design and development of environment-friendly chemical processes. Turbulent combustion of gaseous fuels has been numerically investigated by many researchers for the past six decades using various turbulent models. Mare et al [4] observed that accurate description of the inlet section of the combustor plays a significant role in the prediction of temperature distribution over the flow region in a can-type model gas turbine combustor using numerical simulation technique. Khalil et al [5] suggested that swirl in the combustor will improve the performance with improved fuel mixture preparation prior to the ignition of fuel at higher operational pressures. Decreasing the equivalence ratio changes the flame shape [6] due to reduced flame speed such that the flame fronts overlap the convecting Kelvin–Helmholtz vortices along the shear layer. Ziani et al [7] compared the three turbulence model for non-premixed combustion and evaluated with the numerical results of the literature and experimental data. The evaluation resulted in the conclusion that the modified k-ε model is the most appropriate for simulating the non-premixed turbulence combustion.

The bluff body stabilized flames have found wide range of applications because of their relatively low-pressure drop and simple geometry [8]. The combustion with a bluff body stabilized fame of gases fuel for both the premixed and non-premixed flame conditions have also been numerically and experimentally investigated by many researchers. Giacomazzi et al [9] studied the combustion in a bluff-body flame anchored in a straight channel and the corresponding results for both cold and reactive flows using eddy dissipation concept for a turbulence intensity of 3 to 4 % and an equivalence ratio of 0.65. The impact of increased reactant temperature on the dynamics of bluff-body stabilized premixed flows was investigated by Erickson et al [10]. Vervisch [11] reviewed the use of numerics to understand and model non-premixed turbulent flames and first gave a rapid overview of non-premixed turbulent combustion modelling. Schefer et al [12] measured the velocity in a turbulent bluff-body stabilized methane flame. The flow configuration consisted of a 5.4-mm diameter fuel jet separated from an outer, annular-air flow by a 50-mm diameter bluff-body. For premixed combustion systems, recent studies reported that adding a bluff-body stabilizer within a micro-channel can significantly extend the blowoff limits [13–15], in a manner similar to large-scale gas turbine combustor applications. Wana et al [16] investigated numerically and experimentally the effect of equivalence ratio on combustion characteristics of premixed hydrogen/air flame in a micro-combustor with a bluff body using Fluent 6.3.

While a bluff-body has a desirable effect of flame stabilization by creating a recirculation zone, it may also pose adverse effects on flame stability by the unsteady vortex shedding into the flow field [17–18], causing fluctuations in the region of flame stabilization. Therefore, it is of fundamental and practical interest to understand the detailed physical mechanism of bluff-body flame stabilization within a narrow channel premixed combustor.
The main objective of the present work is to identify a correct model for a geometry under study and then analyse the effects of the various parameters on the overall efficiency of the system. We study the interaction between the flame structure, the flow field and the coupled heat transfer with the flame holder of a CH_4 /air flame stabilized on a heat conducting bluff body in a channel. The impact of the bluff body shape on the flame properties is particularly investigated.

2. **NUMERICAL METHOD**

2.1. Geometry and flow configuration

The geometry of the combustion chamber is shown in Figure 1. The configuration used here is the bluff-body stabilized CH_4 flame investigated experimentally at the Sandia National Laboratories and the University of Sydney documented in [1, 19]. It has an outer diameter of 50 mm with a concentric fuel jet of diameter of 3.6 mm. The complete burner assembly was housed in a cylindrical co-flow wind tunnel of diameter 254 mm. For the flame the fuel jet velocity was 120 m/s with a coflow velocity of 40m/s.

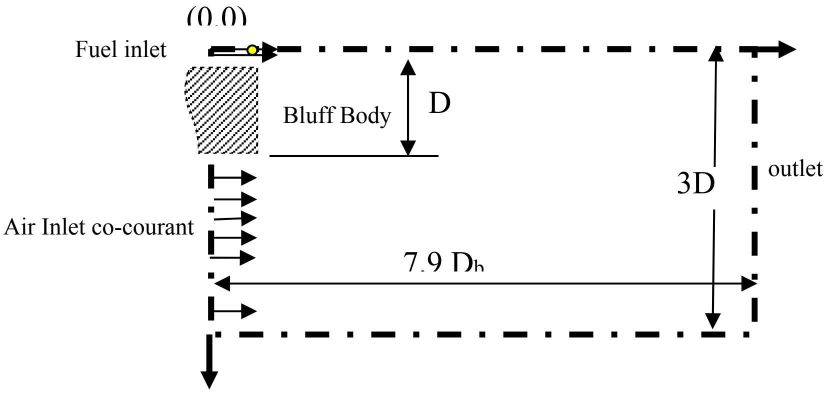


Figure 1 Schematic of the combustion chamber (D_b=50mm and $Ø_{fuel\ inj}$=3.6 mm)

2.2. Computational modelling

The modelling of the geometry was done using Ansys ICEM CFD. Since the geometry is cylindrical without any swirler or tangential velocity inlet, the model can be assumed to be an axi-symmetric problem. The computational domain of the axi-symmetric model of the combustor with boundary conditions is shown in Figure 1.

A fine mesh is imposed around the hollow cylindrical bluff body for precise measurements of the flow conditions (Figure 2). Zero velocity impermeability boundary conditions are specified at the bluff body walls. The boundary conditions are also presented in Figure 1.

The top edge has been taken as wall, the left edge as velocity inlet, the right edge as pressure outlet and the bottom edge as axis. The velocity inlet 1 is for the fuel inlet and the velocity inlet 2 is for the air inlet. A total of 20236 nodes and 104876 elements were used for meshing the geometry. Map-type rectangular meshes with an interval size of 0.2 mm were used.

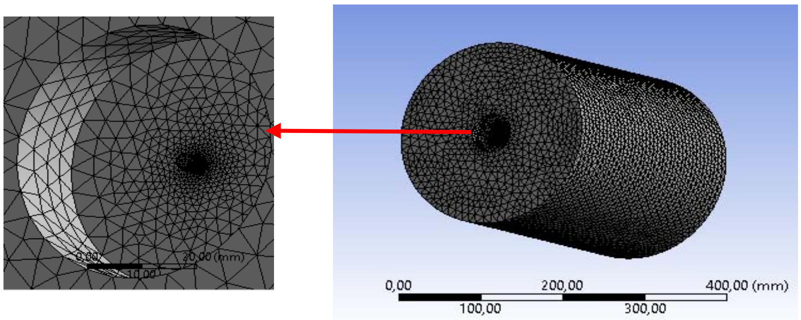


Figure 2. Non uniforme Mesh of the combustion chamber (104876 elements)

2.3. Numerical model validation

The wall material was steel and the surface reaction effects were eliminated. In computational domain, the atmosphere inside the combustor is considered as a fluid and the bluff body shapes are subtracted from the mentioned domain. It means in the solution process the bluff body is assumed as a shape which fluid cannot flow through it. Based on this assumption, the material of bluff body and heat conduction model in this section is not considered and the main discussion is about the whole of micro-combustor.

Besides, compared to the results of ref [12, 19] which thermal conjugate was applied for bluff body, our results could be acceptable.

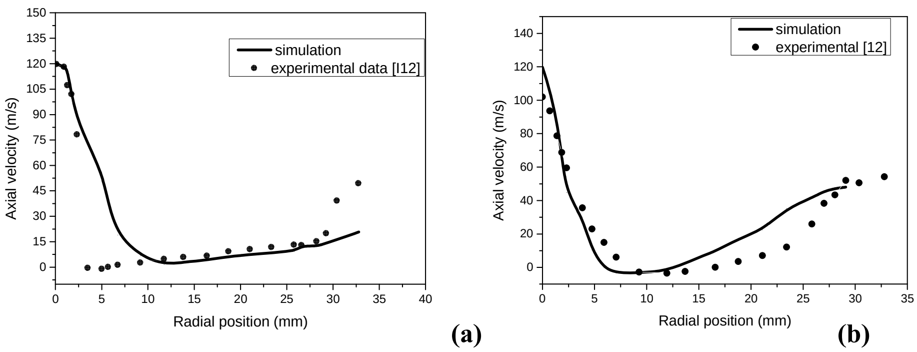
Ansys Ansys CFX 12 was use to solve the momentum equations, energy, species, mass and heat transfer in different states. The first-order upwind scheme was applied for discretization and for the pressure-velocity coupling, the SIMPLE Algorithm was used. Three turbulent models namely, the Renormalized group (RNG) k-ε (2 equations) model, the Shear-stress transport (SST) k-ω (2 equations) model and the Quadratic pressure-strain Reynolds Stress Model (RSM) are proposed by le code. The K-Epsilon model has been used in this work.

Indeed, gradient is set least squares cell based, pressure is standard, momentum and energy are second order upwind and turbulent kinetic energy and turbulent dissipation rate are set first order upwind. The structured square grid system with around 100,000 cells for all cases was set in the final simulation. Three step refinements were

applied to increase the quality of the meshes. In all cases the initial guess for fuel-air mixture was set at 1600 K to ignite the mixture in numerical iteration. The simulations were
done by a core i5 computer with 4 GB of RAM and the cases solved.

2.4. RNG k-ε model

The simplest “complete models” of turbulence are the two-equation models in which the solution of two separate transport equations allows the turbulent velocity and length scales to be independently determined. The RNG model has an additional term in its ε equation that significantly improves the accuracy for rapidly strained flows. The effect of swirl on turbulence is included in the RNG model, enhancing accuracy for swirling flows. The corresponding plots obtained using this model has been shown in Figure 3.



(a) **(b)**

Figure 3. Comparison between numerical and experimental results (a) at 3mm & (b) 30mm from Bluff Body

3. **RESULTS AND DISCUSSIONS**

The numerical simulations have been done using ANSYS CFX 12.0 for all the cases involved in this study. The results comprises of 4 sections. In the first section, the model has been analysed for various values of the velocity ratio.

In the subsequent sections, the analysis was done for various values of the velocity and equivalence ratio. In the final section, a comparative study had been done for two shapes of a bluff body with the same frontal area.

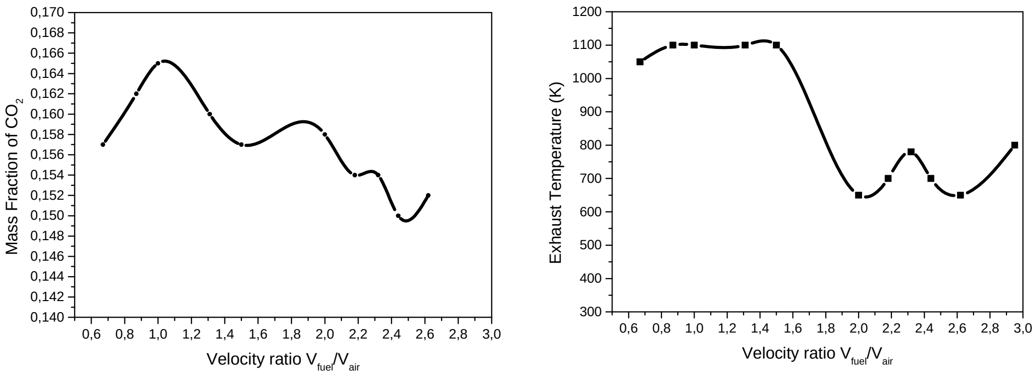
A. Variation in the inlet velocity ratio

This section presents the exhaust temperature and CO2 results for the combustion process of methane and air at an equivalence ratio of 0.5. The case details with different velocity values for the fuel and air has been shown in Table 1 below.

Table 1 Case details with varying inlet velocity ratios

Case	Inlet Air velocity (m/s)	Inlet Fuel velocity (m/s)
1	40	120
2	50	100
3	90	60
4	100	50

The average exhaust gas temperature and the mass fraction results for the cases have been summarized in the graph shown in Figure 4.



(a) **(b)**

Figure 4- Evolution of CO2 mass fraction and exhaust temperature in function of velocity ratios at an equivalence ratio of 0.5

The above result shows that the exhaust gas temperature decreases first and then increases with the decrease of velocity ratio. The lowest temperature is reached at a velocity ratio of 2 and the value was found out to be 678K. Below this point, a uniform temperature of about 1000K is obtained. Thus the optimal combustible velocity ratio is found out to be within 0.6 and 1.8.

The mass fraction of CO2 is also seen to be comparatively higher at lower velocity ratios; the maximum being 0.165 at a velocity ratio of 1. This clearly shows that the difference in velocity between the fuel and the air inlets should be low for higher exhaust temperatures and for a better efficient combustion process.

B. Variation in the velocity magnitude

This section presents the average exhaust gas temperature results at an equivalence ratio of 0.5 and with varying values of the velocity magnitude at a velocity ratio of 1. The velocity values were varied over a range from 60m/s to 110m/s. Table 2 summarizes the cases and results.

The results show that the average exhaust temperature increases first and then decreases as the velocity increases. The temperature was higher over the range of 70m/s to 95m/s and reaches a maximum of 1153K at 80m/s.

C. Variation in the equivalence ratio

This section presents the average exhaust gas temperature results for different velocities and equivalence ratios at a velocity ratio of 1. The velocity values were similar to the previous analysis with equivalence ratio of 0.4, 0.5 and 0.6 simulated for all the cases. The graph shown in Figure 5 depicts the results.

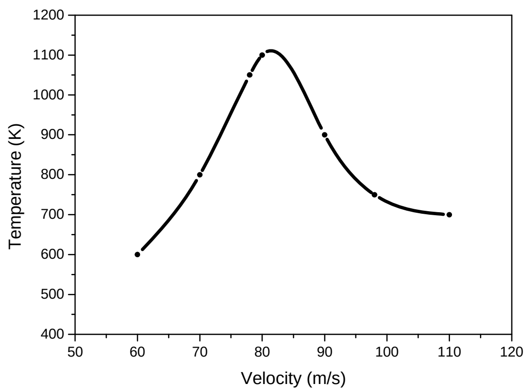


Figure 5. Values of temperature results obtained with different velocities at an equivalence ratio ϕ= 0.5

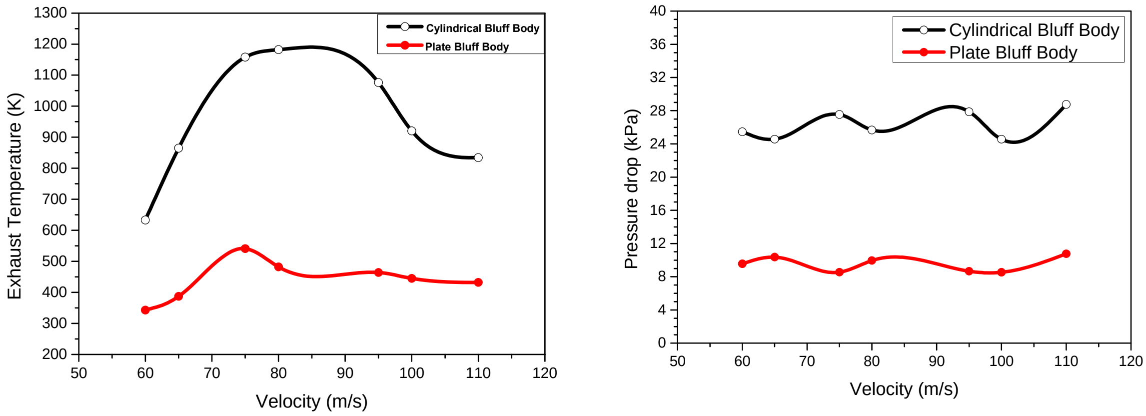
The above graph shows the exhaust gas temperature values for equivalence ratio of 0.5. The result shows that comparatively high combustion efficiency can be maintained over the whole equivalence ratio range at a moderate velocity magnitude. The maximum temperature reached for an ER of 0.5 is 1153K at 80m/s..

D. Variation in the bluff body shape

This section presents the average exhaust gas temperature and pressure drop results for the two bluff body shapes namely (a) circular plate and (b) cylindrical solid and the results are shown in Figure 6.

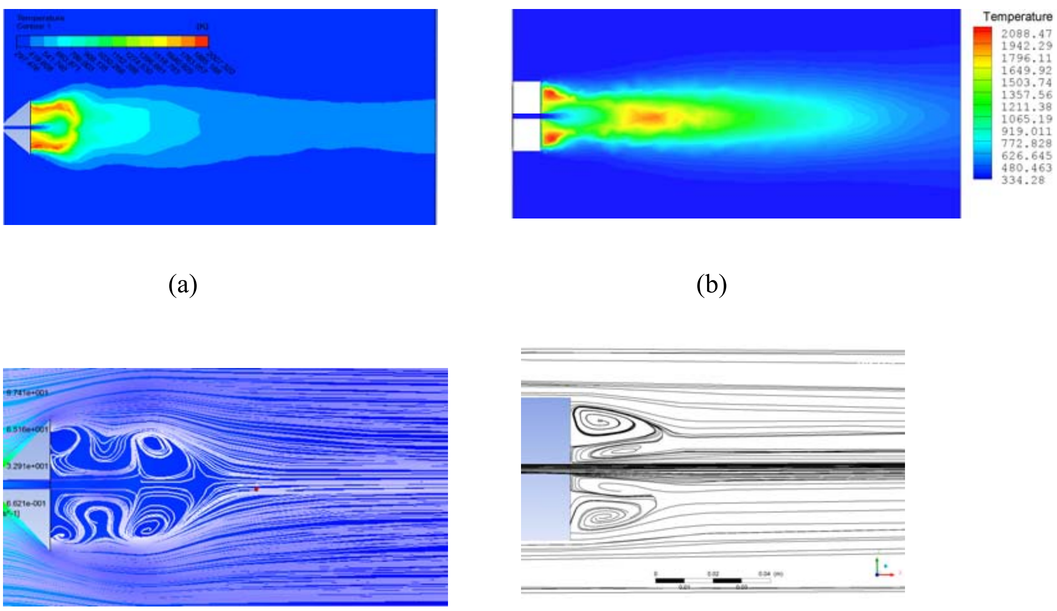
The results show that combustion with cylindrical solid bluff body produces temperatures that are almost double to that produced by the circular plate. This is seen by the enormous pressure drop across the solid bluff body which is of the order of 25KPa compared to the 10KPa drop seen around the circular plate.

Figure 7 and 8 show the streamlines and temperature contour for two bluff body shapes: cylindrical and triangular.



(a) **(b)**

Figure 6. Evolution of Exhaust Temperature (a) and pressure drop (b) with varying inlet velocity values for circular plate and solid cylinder as bluff body



(a) (b)

(c) (d)

Figure 7. Effect of the shape of the bluff body on the temperature contour and streamlines Triangular BB (a and c) , Cylindrical BB (b and d)

4. **CONCLUSION**

Combustion characteristics of lean methane – air mixture in a combustor with a bluff body stabilized non-premixed flow regime were investigated through simulations which were validated with the experimental results. Effects of the inlet velocity ratio, velocity value, equivalence ratio and the bluff body shape on the combustion efficiency and exhaust gas temperature were examined. The following conclusions have been drawn from this study.

1) It was seen that the RSM turbulent model showed better results compared to the other RANS models for the velocity vectors at critical regions of the flow regime which was necessary for further simulation analysis.

2) This study also reveals that, combustion with cylindrical solid bluff body produces temperatures that are almost double to that produced by the circular plate but with significantly higher pressure drop which is the key driver for drag force.

3) Then it was seen that the inlet velocity ratio and the velocity magnitude also exerts a significant effect on the combustion characteristics. Decreasing the velocity ratio initially shows moderate variation in the temperature.

After a particular stage, there seems to be higher temperature values when the ratio is near unity i.e. when the air and fuel are at nearly the same velocity magnitude. Furthermore, with the increase of inlet velocity, the high temperature zone is pushed toward the downstream of the combustor chamber. However, the exhaust gas temperature reaches a peak value at a moderate inlet velocity under a fixed equivalence ratio.

These variation tendencies indicate that the combustion characteristics are in fact determined by many factors like disorderliness, heat losses, recirculation zones, momentary pressure drops, turbulence and their complicated interactions.

REFERENCES

[1] Dally B B , Fletcher D F & Masri A R (1998) Flow and mixing fields of turbulent bluff-body jets and flames, Combustion Theory and Modelling, 2:2, 193-219.

[2] Senda M, Sawada N and Tsuchiya M, (1993), A numerical prediction of turbulent non premixed flame stabilised by a bluff body with laminar flamelet model, *9th Symp. on Turbulent Shear Flows (Kyoto)* p 103.

[3] Masri A R, Dally B B, Barlow R S and Carter C D, (1994), The structure of the recirculation zone of a bluff-body combustor, *25th Int. Symp. on Combustion* (Pittsburgh, PA: Combustion Institute) pp 1301-8.

[4] Mare F, Jones WP and Menzies KR, Large eddy simulation of a model gas turbine combustor, Combustion and Flame 137 (2004) 278–294

[5] Khalil A, Gupta AK, Distributed swirl combustion for gas turbine application, Applied Energy 88 (2011) 4898–4907

[6] Tuttle SG, Chaudhuri S, Kopp-Vaughan KM, Jensen TR, Cetegen BM, Renfro MW, Cohen JM, Lean blow off behaviour of asymmetrically bluff body stabilized flames, Combustion and Flame (2013)

[7] Ziani L, Chaker A, Chetehouna K, Malek A, Mahmah B, Numerical Simulations of non-premixed turbulent combustion of CH4 – H2 mixtures using the PDF approach, Hydrogen Energy (2012)

[8] Boudier G, Gicquel LYM, Poinsot T, Bissie`res D, Be´rat C, Comparison of LES, RANS and experiments in an aeronautical gas turbine combustion chamber, Proceedings of the Combustion Institute 31 (2007) 3075–3082

[9] Giacomazzi E, Battaglia V, Bruno C, The coupling of turbulence and chemistry in a premixed, Combustion and Flame 138 (2004) 320–335

[10] Erickson RR, Soteriou MC, The influence of reactant temperature on the dynamics of bluff body stabilized premixed flames, Combustion and Flame 158 (2011) 2441–2457

[11] Vervisch L, Using numerics to help the understanding of non-premixed turbulent flames, Proceedings of the Combustion Institute, Volume 28, 2000/pp. 11–24

[12] Schefer RW, Namazian M and Kelly J., “Velocity Measurements in a Turbulent non-premixed Bluff-Body Stabilized Flam” Combustion Science and Technology, Volume 56, Issue 4-6, 1987, pp. 101-138

[13] A. Fan, J. Wan, K. Maruta, H. Yao, W. Liu, Int. J. Heat Mass Transfer 66 (2013) 72–79.

[14] A. Fan, J. Wan, Y. Liu, B. Pi, H. Yao, K. Maruta, W. Liu, Int. J. Hydrogen Energy 38 (2013) 11438–11445.

[15] G. Bagheri, S.E. Hosseini, M.A. Wahid, Appl. Thermal Eng. 67 (2014) 266–272.

[16] Wana J, Fan A, Maruta K, Yao H, Liu W, Experimental and numerical investigation on combustion characteristics of premixed hydrogen/air flame in a micro-combustor with a bluff body, international journal of hydrogen energy 37(2012)19190-19197

[17] H. Suzuki, Y. Inoue, T. Nishimura, K. Fukutani, K. Suzuki, Int. J. Heat Fluid Flow 14 (1993) 2–9.

[18] J. Hertzberg, I. Shepherd, L. Talbot, Combust. Flame 86 (1991) 1–11.

[19] Dally BB, Masri AR, Barlow RS, Fiechtner GJ, Instantaneous and mean compositional structure of bluff-body stabilized non premixed flames. Combustion and Flame, 1998; 114:119-48