Flowfield features of cyclonic patterns for MILD Combustion technologies

G. Sorrentino[1], M. Agrillo[2], R. Ragucci[2], C.E. Frouzakis[3]

1. *DICMAPI-Università degli Studi di Napoli Federico II, P.le Tecchio 80, 80125, Naples, Italy*
2. *Istituto di Ricerche sulla Combustione-CNR, P.le Tecchio 80, 80125, Naples, Italy*
3. *Aerothermochemistry and Combustion Systems Laboratory, Swiss Federal Institute of Technology Zurich, CH-8092 Zurich, Switzerland*

Introduction

MILD combustion [1, 2] is a combustion regime characterized by fuel oxidation in an atmosphere with relatively low oxygen concentration and high inlet temperatures featuring a process with a distributed reaction zone, relatively uniform temperatures within the combustion chamber, no visible flame, and low pollutant emissions [3].

Existing industrial MILD Combustion systems usually reach the autoignition conditions by recirculating efficiently the product gases into the incoming fresh reactants [4, 5]. The exhausted gas recirculation serves two purposes: (i) to raise the reactant temperature (heat recovery) and (ii) to reduce the oxygen concentration (dilution). To realize an efficient MILD combustion process the establishment of an effective mixing process between the recycled gases with the fresh reactant jets can play a very important role. This in turn produces locally high turbulent mixing rates to inhibit oxidation reactions before reaching diluted conditions [6]. Another key point for MILD systems is the realization of relatively long residence time within the combustion chamber because dilution levels imply longer kinetic characteristic times with respect to traditional systems [7, 8].

In summary, the entity of recycled heat and mass, the efficiency of mixing and relatively long residence times are key factors in the establishment of MILD combustion conditions. The recirculation of hot reactive species into the fresh stream can be achieved through proper design of the combustor flow field. Flow entrainment can be obtained with different flow arrangements. It can be achieved through internal or external recirculation; however, internal recirculation is favorable for many applications. One common practice used to create entrainment and stabilize combustion involves the use of cyclone-type configuration [9], where the recirculation of hot active species is achieved through two key features: main jet entrainment, and recirculation of gases due to the geometry of the combustor itself.

In this paper, the non-reacting flow field of a cyclonic configuration is examined with focus on entrainment and recirculation generation in order to achieve flameless distributed combustion and enhance thermal and environmental performance of the combustor with ultra-low emissions. A non-reacting flow field is considered in order to study the flow characteristics inside the combustor along with the turbulence generated and the mixing characteristics.

In this context, the cyclonic flow pattern inside the chamber is studied and in particular the velocity field is studied through PIV and numerical analysis. Inlet Reynolds number of Re=1000 was chosen as a representative test-case for experimental and numerical comparison.

Experimental and Numerical methods

In order to study the non-reacting flow field of a cyclonic combustion chamber, a test chamber was built and characterized experimentally using particle image velocimetry (PIV) under isothermal conditions in terms of the behavior of the jets and the assessment of the cyclone structure inside the chamber for a fixed value of the inlet jet Reynolds number. Measurements

corresponding author: *Dr. Giancarlo Sorrentino , email: g.sorrentino@unina.it*

were carried out in a glass windowed chamber, designed to operate at atmospheric pressure and environmental temperature conditions in order to achieve a full optical access of the device and to simplify maintenance and cleaning operations. The apparatus consists of a square section (0.2x0.2m), in order to minimize any reflections of the laser light sheet, a height of h=0.05 m, inlet jets diameter d_{jet}= 0.008 m, and a section exit placed on the top side, with a diameter d_{out}= 0.025 m. In addition to the test chamber, the experimental system consists of an optical detection apparatus, gas supply lines and a tracer insemination system for the flow visualization.
The optical characterization of the chamber is performed by recording the pattern of the light elastically scattered when a laser sheet illuminates the tracer. A Nd:YLF pulsed laser was tuned on the second harmonic wavelength (λ = 527 nm) and its beam was shaped by a set of cylindrical lenses to a sheet of constant thickness. It was varied in height by the extension of the objective field. Patterns of elastic scattered light was detected by a CMOS camera with a variable-focus telescope. Since each pulse is in a different frame, there is no directional ambiguity for the velocity vectors. PIV measurements were conducted on the global field of view 200x200 mm for obtaining information on the flow-field developed into the chamber. All the acquisitions were carried out on the middle plane of the apparatus.
Seeding tracer was feed in only one of the two inlet jets. Helium at 300 K was used as working fluid in order to reproduce high-temperature conditions (i.e. similar properties of CO_2 at 1000 K). PIV Images were acquired at a rate of 1 kHz and the digitized images were cross-correlated using a recursive rectangular grid algorithm using 32 x 32 pixel and then 16 x 16 pixel interrogation windows to find the mean pixel displacement.
Numerical simulations were carried out in order to characterize the fluid dynamic pattern using three turbulent RANS models implemented in ANSYS Fluent, in order to compare the results and observe their differences: k-ε RNG model, k-ω SST model and a 7-equation Reynolds Stress Model. The inlet velocity that was chosen for the comparison were: v_{inlet}=14 m/s, corresponding to a Reynolds number of $Re_{jet}=v_{inlet}\, d_{jet}/\nu=1000$. A SIMPLE resolution scheme was selected, while for the momentum equation was set a second-order upwind scheme.

Results

In the following are reported the results obtained with the PIV and numerical simulations of the cyclonic reactor for a Reynolds number of 1000 for the inlet Helium jet.
Firstly, the experimental contours of velocity magnitude will be examined, as reported in Figure 1, in order to give a global idea of the characteristic velocities of the cyclonic flow field.

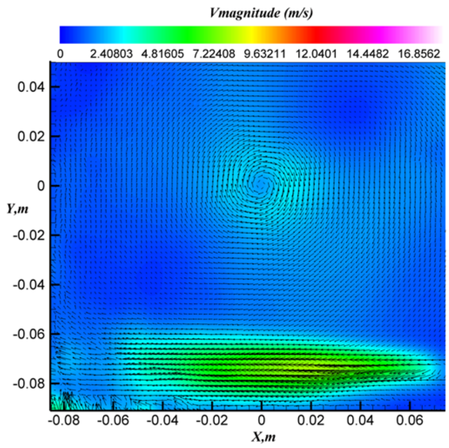

Fig. 1 Mean Velocity Vectors for Re = 1000

The experimental campaign showed that the jet behavior depends strongly on Re. the collected data highlighted that the interaction of the jets leads to a cyclonic vortex structure and it shows a different characteristic behavior respect to a free jet in terms of velocity decay, jet width and

fluid-dynamic structure. Such a configuration demonstrates a high recirculation level throughout the combustor, critical to achieve distributed combustion conditions.
The flow-field is also characterized by the presence of a high velocity region at the outer boundary with another high velocity region near the center. This behavior is almost consistent with reported velocity behavior in cyclone combustor [10,11]. A detailed description of the flow pattern in a cyclone is given in the literature [9, 12], where two flows rotating coaxially and carrying the main mass of gas were observed: the wall flow and the center flow, separated by a zone occupied by the increasing turbulent vorticity branching from those two flows.
The characteristic profiles of tangential velocities in the chamber volume, which were measured in the midsection along the Y=0 line are compared in Fig. 2 with the simulation results The tangential velocity has a typical maximum and three flow regions: the region in the center of the chamber, the region of potential rotation on the periphery of the chamber and a wall region.
Tangential velocity profiles obtained with the numerical simulation were compared, for the case at Re_{jet}=1000, with the ones obtained with the experimental tests over a line.

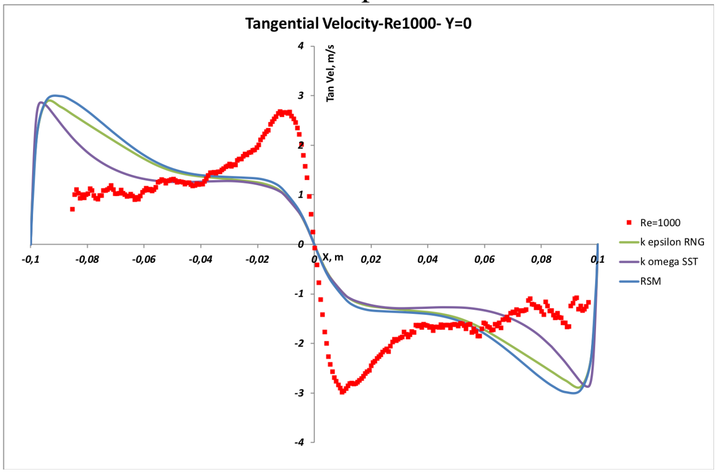

Fig. 2 Comparison of experimental and numerical tangential velocities along X in the midsection for Y=0 and Re_{jet}=1000

As can be observed, the numerical and the experimental results are close to each other only in the peripheral region of the cyclone (0.03 m<X<0.08 m). The disagreement in the central core zone is probably due to a limitation of the turbulence models. None of the RANS models can predict the very strong gradients, in the direction normal to the outflow, that characterize the reverse-flow zone of the cyclone and which are related to the Reynolds Stress Tensor. The most accuracy that is available is the RSM, is still not sufficient to describe properly the flow field in that region.
Despite its higher precision, compared to the 2-equation model, it does not return reliable results. On the other hand the k-ω SST model is in better agreement with the experimental results, especially for 0.04 m<X<0.07 m.
In this context, the possibility of using scale-resolving turbulence models (LES or DNS approach) could represent an important step forward to enhance the understanding of unsteadiness and flow-structures inside the chamber.

Conclusions and Future perspectives

Application of the MILD combustion concept requires careful examination on the role of mixing process between fresh air-fuel mixtures with the recirculated hot products, to prepare lean and ultra-lean mixtures. In this study, the velocity pattern of the isothermal flow field in a cyclonic chamber was studied experimentally and numerically. The flow field exhibited high recirculation ratio, which enhances distributed combustion. High recirculation ratio is critical for distributed combustion as the entrainment of hot product gases is essential to form reactive and diluted oxidizer when combined with fresh air stream. The flow field was investigated at a fixed inlet jet Reynolds number of 1000 and the tangential velocity patterns have given an overall description of the cyclone structure.

The high velocity region near the center exhibits a non-monotonic trend with the cyclone radius.In the peripheral region of the chamber, the tangential velocity maintains almost constant values and in this region the comparison between PIV and RANS is satisfactory whereas scale-resolving turbulence models seems to be necessary in the core and close to wall regions.

References

[1] Cavaliere, A., de Joannon, M. Mild combustion, Prog. Energy Combust. Sci. 30 (2004) 329-366.

[2] Li, P., Mi, J., Dally, B. B., Wang, F., Wang, L., Liu, Z., Chen S., Zheng, C. Progress and recent trend in MILD combustion. Science China Technological Sciences, (2011), 54(2), 255-269.

[3] de Joannon, M., Sorrentino, G., Cavaliere, A. MILD combustion in diffusion-controlled regimes of hot diluted fuel. Comb. Flame, (2012), 159(5), 1832-1839.

[4] Abtahizadeh, E., van Oijen, J., de Goey, P. Numerical study of mild combustion with entrainment of burned gas into oxidizer and/or fuel streams. Comb. Flame, (2012), 159(6), 2155-2165.

[5] Rafidi, N., Blasiak, W. Heat transfer characteristics of HiTAC heating furnace using regenerative burners. Appl. Therm. Eng., 26 (2006), pp. 2027–2034

[6] Khalil A.E.E., Gupta A.K. Distributed Swirl Combustion for Gas Turbine Application. Applied Energy, (2011), 88: 4898-4907.

[7] Sabia, P., de Joannon, M., Lavadera, M. L., Giudicianni, P., Ragucci, R. Autoignition delay times of propane mixtures under MILD conditions at atmospheric pressure. Comb. Flame, (2014), 161 (2014) 3022-3030.

[8] Sabia, P., Lavadera, M.L., Giudicianni, P., Sorrentino, G., Ragucci, R., de Joannon, M. CO2 and H2O effect on propane auto-ignition delay times under mild combustion operative conditions. Comb. Flame, (2014), 162, (3), 533–543.

[9] A.K. Gupta, D.G. Lilley, N. Syred, Swirl flows. Abacus Press, Tunbridge Wells, England, UK (1984).

[10] E.D. Baluev, Y.V. Troyankin, The effect of the design parameters on the aerodynamics of cyclone chambers. Thermal Eng, 14 (2) (1967), pp. 67–71

[11] B.P. Ustimenko, M.A. Bukhman. Turbulent flow structure in a cyclone chamber. Thermal Eng, 15 (2) (1968), pp. 64–67

[12] Pitsukha, E. A., Teplitskii, Y. S., Borodulya, V. A., Volchkov, E. P., Dvornikov, N. A., & Lukashov, V. V. (2012). Investigation of the structure of swirling flows in a cyclone chamber under various conditions of gas inlet and outlet. Journal of Engineering Physics and Thermophysics, 85(2), 324-338.