SHORT TERM SCIENTIFIC MISSION (STSM) – SCIENTIFIC REPORT

Direct numerical simulation (DNS) of cyclonic flow-fields

Home Institution:

The Institute for Research on Combustion (IRC)-CNR

Host Institution:

ETH Zurich, Department of Mechanical and Process Engineering, Institute of Energy Technology

Action number: CM1404
STSM title: SMARTCATS
STSM start and end date: 22/10/2018 to 5/11/2018
Grantee name: Marco Agrillo

PURPOSE OF THE STSM

Moderate or Intense Low oxygen Dilution (MILD) combustion [1] is one of the most promising new technologies in the area of the high-efficiency and sustainable chemical processes. The interaction between fluid dynamics and kinetics is very strong under MILD combustion conditions: chemical and local diffusion time scales are coupled on the macroscopic levels, nevertheless, there are two opposite effects: flame stabilization is favored by local conditions with low velocities, gaseous jets mixing is favored by high velocities and high turbulence rates. The way to satisfy these requirements in order to reach a MILD and stable combustion regime, is through an internal recirculation of exhaust gases.

There are many practical ways to get an internal recirculation of exhausts, one of the most promising technological applications are cyclonic flows. Cyclonic flows which currently find wide use in gas-solid separators, have attracted a lot of interest for application in combustion technologies such as MILD combustion. With a cyclonic flow, indeed, it is possible to achieve mixing in a very short time, while allowing the flow to have a long residence time inside the reactor [2]. As in many confined swirling flow systems, the hydrodynamic phenomena in cyclones can be very complex. Flow reversal, quasi periodic fluctuations and strongly anisotropic turbulence are some of the characteristics [3].

The main purpose of this activity lies in the advanced numerical simulation of the cyclonic burner [4] built by the IRC-CNR Research Group in Naples. The numerical studies of the system fluid-dynamics will be based on the modelling of the non-reactive and isothermal pattern in the chamber. Preliminary CFD studies using Reynolds-averaged Navier-Stokes (RANS) models were performed on the experimental setup, and they were compared with PIV measurements of the cold flow field. RANS accounts only for the mean fields, and models the turbulence scales and does not resolves unsteady flow structures, it cannot provide for a detailed description of the cyclonic flow field.

In this context, the possibility of using scale-resolving turbulence models is an important step forward to enhance the understanding of unsteadiness and flow-structures inside the chamber. Direct numerical simulation (DNS) resolves all scales of scalar and velocity fluctuations. Through DNS it would be possible to describe the unsteady fluid-dynamic characteristics of the cyclone inside the chamber from both the macro and microscopic point of view. However, DNS carries a very high computational cost, which increases strongly with the domain size. The computational cost required by DNS make the pre-processing step of grid generation very important in order to reduce the overall computational efforts. Therefore, the first part of the study involves the generation of a computational mesh of the cyclonic chamber, adapted to have finer resolution in the regions where small flow structures and high gradients exist.

The aim of the STSM was to establish the collaboration between the ETH Zurich and IRC-CNR and take the first steps towards the direct numerical simulation of the experimental cyclonic burner. The DNS code Nek5000, which is extensively used at ETH will be used to capture the unsteady fluid-dynamic characteristics of the cyclone inside the chamber from both a macro and microscopic point of view. The STSM is of fundamental importance in order to allow the two groups to cooperate for the improvement of modeling of the non-reactive and isothermal pattern inside the cyclonic burner.

DESCRIPTION OF WORK CARRIED OUT DURING THE STSM

The first step was the familiarization of the ETH group with the burner and the PIV experiments that have been performed at IRC-CNR, in order to construct the computational grid. The chamber dimensions are 20×20×5 cm^3 fitted with two nozzles of diameter d_j=0.8 cm and an opening where the flow exits of diameter d_o=2.5 cm (Fig. 1).

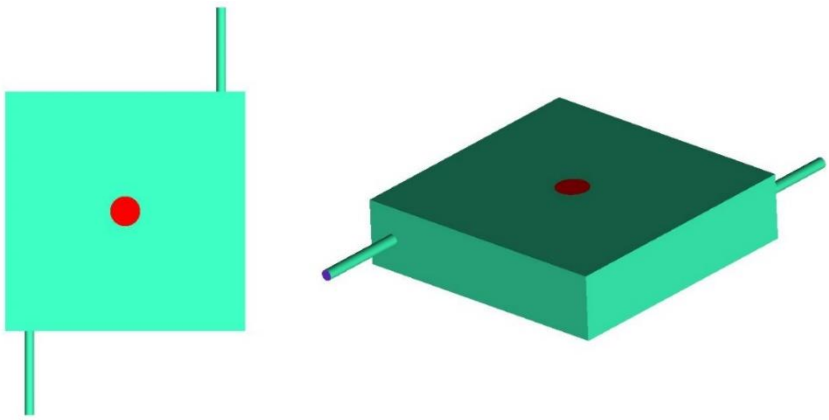

Figure 1 ***Geometric configuration of the cyclonic flow chamber.***

Since the first aim is to investigate the of the isothermal and non-reactive flow inside the chamber, only the oxidant nozzles are present in Fig. 1, unlike the cyclonic burner [4].

The ANSYS module ICEM-CFD was used to generate the mesh. It is a state-of-the-art meshing tool able to generate both structured and unstructured grids. ICEM-CFD was used to generate grids for the first exploratory DNS to assess resolution requirements.

Nek5000 employs purely hexahedral conforming grids, which can be locally refined. Side, top and front views of the one million hexahedral elements are shown in fig. 2.

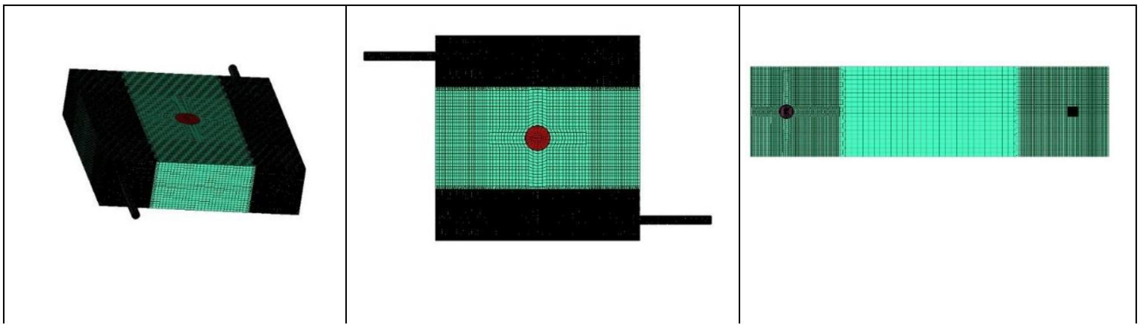

Figure 2: Side, top and front view of the locally-refined cyclonic chamber mesh

The mesh is refined locally in the inlet zone of the chamber and along the direction of jet flow development, where the highest velocity gradients exist and the unstable flow structures are generated.

PRELIMINARY DNS RESULTS

Direct numerical simulations (DNS) were conducted using Nek5000 [8], which employs the spectral element method (SEM) [9], a high-order weighted residual technique similar to the finite element method, to discretize the governing incompressible Navier-Stokes equations.

The condition for the DNS are the ones considered in RANS, where helium at 298.2 K flows with a mean inlet velocity of V_{avg}=16 m/s though both nozzles into the cyclonic burner; the jet Reynols number at these conditions is Re=1,000. Guided by the initial simulations with the grid shown in Fig. 2, a new grid with 1.13 million spectral elements was constructed. The geometry was slightly modified geometry by adding a short extension at the outflow, as shown in Fig. 3 together with three views of the spectral element skeleton. The simulations were performed using 5^{th}-order polynomials to approximate the solution within each spectral element. This corresponds to a total of approximately 244 million grid points with a resolution of 120 μm inside the nozzles and 200 μm in the finer resolution areas close to the walls. Instantaneous isosufarces of the y- velocity component and the distribution of the x- velocity on the horizontal midplane extracted from the simulations are shown in Fig. 4.

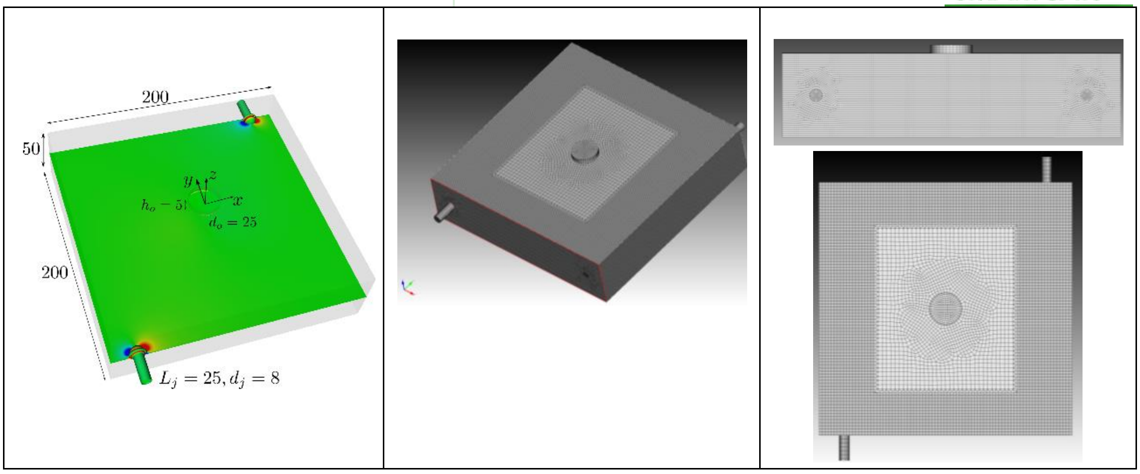

Figure 3. Modified geometry and three views of the refined mesh used in the current DNS.

FUTURE PERSPECTIVES

The exchange of expertise between the two groups will continue with the analysis of the numerical results obtained with the different approaches (RANS, LES, DNS) and comparison with the available experimental data to identify all the unstable fluid-dynamic characteristics of the cyclone inside the chamber. The results will be summarized in a joint publication.

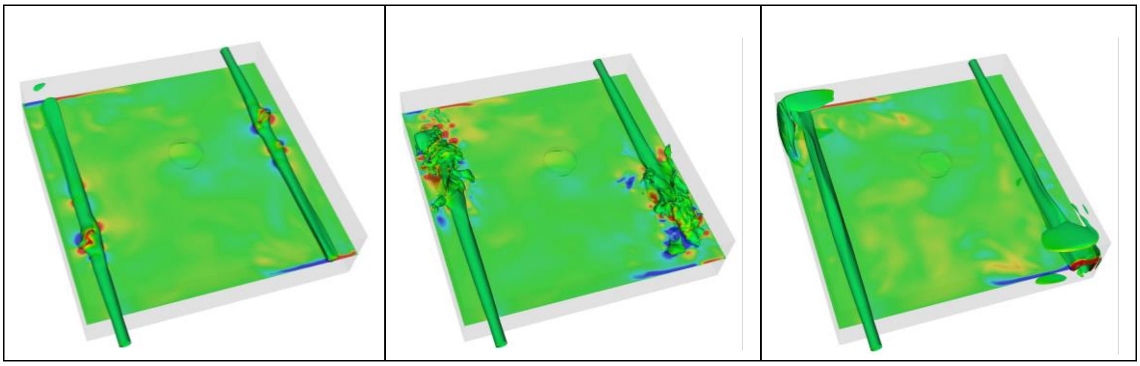

Figure 4. Instantaneous plots of isosurfaces of the v=+/-3.2 m/s y-velocity component superimposed to contour plot of the x-velocity component on the burner midplane.

EUROPEAN COOPERATION IN SCIENCE AND TECHNOLOGY

CM1404 SMARTCATs

References

1. **A Cavaliere, M de Joannon** - *MILD combustion*, Progress in Energy and Combustion science, 2004
2. **M. de Joannon, P. Sabia, G. Sorrentino, P. Bozza, R. Ragucci** - *Small size burner combustion stabilization by means of strong cyclonic recirculation* - Proceedings of the Combustion Institute – 36, 3, 2017, 3361-3369
3. **O Lucca-Negro, T O'doherty** - *Vortex breakdown: a review* - Progress in energy and combustion science, 2001

4. **G. Sorrentino, M. de Joannon, P. Sabia, G. Sorrentino, P. Bozza, R. Ragucci** *Impact of external operating parameters on the performance of a cyclonic burner with high level of internal recirculation under MILD combustion.* Energy ,Volume 137, 15 October 2017, Pages 1167-1174
5. **V. Moureau, P. Domingo, L. Vervisch -** *From Large-Eddy Simulation to Direct Numerical Simulation of a lean premixed swirl flame: Filtered laminar flame-PDF modeling.* Combustion and Flame ,Volume 158, Issue 7, July 2011, Pages 1340-1357
6. **Frisch, U. (1995).** *Turbulence: the legacy of AN Kolmogorov***.** Cambridge university press.
7. **P Moin, K Mahesh,** *Direct numerical simulation: a tool in turbulence research*, Annu. Rev. Fluid Mech. 1998. 30:539–78
8. **Paul F. Fischer, James W. Lottes, Stefan G. Kerkemeier, 2008**, http://nek5000.mcs.anl.gov.
9. **Patera, A.T.** *"A spectral element method for fluid dynamics : laminar flow in a channel expansion."* J. Comput. Phys. **54**:468–488, 1984.