1. INTRODUCTION

High temperature air combustion (HTAC) processes try to control the pollutants formation, in particular NOx emissions, in combustion applications reducing the residence time of gases in high temperature regions of the burner, or avoiding high oxygen concentration in these regions. In the last decade, significant efforts have been made to obtain high thermal efficiencies, due to air-preheating, without the adverse effect of high NOx emissions caused by flames at high temperatures [1,2]. It was found that a strong exhaust gas recirculation combined with air preheating generates relatively low flame temperatures. The combustion air is diluted with a large amount of recirculated exhausts, so the concentration of oxygen within the reaction zone is much lower than in the case of undiluted air allowing for a better control of the furnace temperature. This combustion technology is commonly defined as flameless [3,4] or mild combustion [5-8].

The possibility to reach mild combustion conditions using methane or natural gas as fuels have been extensively studied [3,4,6], as well as the feasibility of mild combustion of hybrid hydrocarbon-hydrogen mixtures [8]; on the other hand, much less information are available concerning the influence of several operating parameters on mild combustion sustainability for these fuels. For this reason, an experimental study has been performed in a laboratory-scale burner, provided by a single high-velocity nozzle, in order to evaluate the influence of different parameters, namely air pre-heating, dilution ratio and equivalence ratio on mild combustion sustainability for methane; then, the nozzle configuration has been changed, using two nozzles to feed separately air and fuel to the burner. It was found that a stable mild combustion can be obtained with a dual nozzle configuration; moreover, once the burner is operated in mild conditions it is possible to change the feeding procedure of the reactants, switching from a single to a dual nozzle configuration, with a negligible influence on both mild sustainability and pollutants emissions.

Two numerical models, one for the single high-velocity nozzle, the other for the combustion chamber, were developed with a commercial CFD code. Several simulations were performed for both flame and mild conditions in order to preliminary evaluate the burner fluid dynamics and evaluating the dilution levels and the interactions between the jets for the different configurations used for the nozzles.

2. EXPERIMENTAL SECTION

A burner, designed and used in previous mild combustion studies [6,8], provided by a single high-velocity jet nozzle has been used in the first part of this work. The experimental apparatus is constituted by the burner, a flow rate control section for the reactants feeding, and a sampling and measurement section for the detection of temperatures and exhaust gases compositions [6]. The lab-scale burner is a vertical quartz tube made by two sections: the combustion air preheating zone (filled by quartz pellets to enhance heat exchange), and the combustion chamber (height =330mm, ID=50mm). The bottom part of the burner is equipped by the reactants inlets, as sketched in figure 1; in particular, case A refers to the well-known
single high-velocity nozzle configuration, into which fuel is fed perpendicularly through a capillary pipe. It is also possible to notice the positions of other inlets, for the incoming flow rates of primary preheated air (A1) and inert gas (N2) and for the secondary air flow rate (A2). This setup has been used to evaluate the influence of the equivalence ratio and to define mild combustion boundaries in terms of average furnace temperature vs. dilution ratio, while the layout shown in case B, which is characterized by a direct injection of the fuel inside the combustion chamber, separated by the air jet, has been used in the second part of the study. In the latter configuration, a stainless steel pipe (OD = 1/16”; ID= 0.2mm) has been used as a secondary injector for pure methane; the fuel jet enters practically to the centre of the combustion chamber, interacting with the incoming air jet. Since the burner can operate both in flame or in mild conditions, it is worth to be noticed that the lateral and separated fuel injections were performed only when the burner had reach stable mild conditions. Gas sampling has been performed by an on-line gas analyzer (HORIBA PG–250), which allows for measuring the concentrations of nitrogen oxides, oxygen, carbon monoxide and carbon dioxide. Moreover, two or three thermocouples have been located at different heights into the furnace to detect temperatures inside the combustion chamber.

![Fig.1 Reactants feeding system: (A) single-nozzle and (B) dual-nozzle layout, respectively.](image)

3. NUMERICAL MODEL

Numerical models of the single-nozzle (SN) and of the combustion chamber were developed with a commercial CFD code. Therefore, a preliminary 3D study was aimed at investigating the interactions between the reactants inside the single-nozzle body, where the fuel enters perpendicularly within the main stream of preheated air. An hybrid computational grid was used. The capillary pipe for the CH₄ feed was meshed with tetrahedral elements, whereas the remainder of the nozzle was modeled with hexahedral elements; after a preliminary grid independency study, a total of about 200k elements was used. Since this nozzle does not have an axial symmetry, the main aim of these simulations was to evaluate the proper distribution of the reactants concentrations, velocity and temperature at the nozzle tip; this allows to study the degree of the partial premixing between fuel and air for different feeding conditions and if a reaction can occur into the nozzle body. Moreover, the profiles of species, velocity, turbulence intensity, temperature etc, estimated at the nozzle tip, were then applied as boundary conditions to the burner inlet during the combustion chamber simulations. The presence of the 3 eccentric exits indicates that the computational domain, for the combustion chamber, should be made at least of a 120° angular sector of the burner; actually, the possible presence of a lateral inlet for the fuel in the dual nozzles (DN) configuration, as well as the symmetry with respect to a vertical plane showed by the profiles computed in the single-nozzle study, suggested to use a computational domain representing a half burner. For this model two tetrahedral meshes were found suitable: 300k elements were used for the study of SN cases and 515k for DN cases (due to the needing to reproduce also the weak jet of fuel.
entering from the side wall of the furnace.

Favre-averaged Navier-Stokes equations were solved across the domain. A modified version of the k- turbulence model was used, as such model is known to over-predict the spreading and decay rate of a round jet flow [9]. In this model the constant C1 of the dissipation transport equation is set equal to 1.6 instead of 1.44, as this was found to improve the accuracy of the k- model in predicting self-similar jets [9]. A combined Finite Rate Chemistry/Eddy Dissipation (FR/ED) Model was chosen to take into account turbulence-chemistry interaction as this model appears suited in particular for mild combustion conditions. As previously mentioned, the high dilution levels and the relatively low temperatures slower the chemical rates making them comparable to turbulent mixing phenomena, which are enhanced by recirculation. Therefore the turbulence-chemistry interaction treatment becomes a crucial point in the modeling procedure [10]. In this preliminary study, the oxidation of the methane fuel was modeled with a single-step mechanism [11] as the main goal was to provide information concerning the flow patterns and expected internal recirculation regions which should be characteristic of different operating conditions and combustion regimes.

As a consequence, simulations were performed for different feeding conditions of combustion air and CH4, in order to replicate several representative experimental situations. Details of the feed streams are reported in table 1 for the nozzle simulations and in table 2 for the burner ones, respectively; concerning table 2, cases 1 and 3 refer to flame operations, whereas runs 2, 4 and 5 refer to mild conditions. For the former cases, secondary air was injected coaxially to the single fuel/air injection nozzle.

### Table 1 Summary of simulated runs: single nozzle (SN).

<table>
<thead>
<tr>
<th>run</th>
<th>Combustion model</th>
<th>FCH4,SN [g/s]</th>
<th>FAir1 @ Tp [g/s]</th>
<th>Temp. preheat (Tp) [K]</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>FR/ED</td>
<td>2.391e-3</td>
<td>1.925e-2</td>
<td>1300</td>
</tr>
<tr>
<td>2</td>
<td>FR/ED</td>
<td>5.953e-3</td>
<td>1.078e-1</td>
<td>1300</td>
</tr>
<tr>
<td>3</td>
<td>FR/ED</td>
<td>3.977e-3</td>
<td>6.665e-2</td>
<td>1300</td>
</tr>
<tr>
<td>4</td>
<td>FR/ED</td>
<td>2.752e-3</td>
<td>9.874e-2</td>
<td>1300</td>
</tr>
</tbody>
</table>

### Table 2 Summary of simulated runs: combustion chamber.

<table>
<thead>
<tr>
<th>run</th>
<th>Mode</th>
<th>Feed layout</th>
<th>FCH4,SN [g/s]</th>
<th>FCH4,DN [g/s]</th>
<th>FAir1 @ Tp [g/s]</th>
<th>FAir2 [g/s]</th>
<th>Temp. preheat (Tp) [K]</th>
<th>Wall temp. [K]</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Flame</td>
<td>SN</td>
<td>2.391e-3</td>
<td>-</td>
<td>1.925e-2</td>
<td>6.842e-2</td>
<td>1300</td>
<td>1000</td>
</tr>
<tr>
<td>2</td>
<td>MILD</td>
<td>SN</td>
<td>5.953e-3</td>
<td>-</td>
<td>1.078e-1</td>
<td>-</td>
<td>1300</td>
<td>1000</td>
</tr>
<tr>
<td>3</td>
<td>Flame</td>
<td>SN</td>
<td>3.977e-3</td>
<td>-</td>
<td>6.665e-2</td>
<td>2.622e-2</td>
<td>1300</td>
<td>1100</td>
</tr>
<tr>
<td>4</td>
<td>MILD</td>
<td>DN</td>
<td>2.752e-3</td>
<td>2.752e-3</td>
<td>9.874e-2</td>
<td>-</td>
<td>1300</td>
<td>1100</td>
</tr>
<tr>
<td>5</td>
<td>MILD</td>
<td>DN</td>
<td>-</td>
<td>5.504e-3</td>
<td>9.874e-2</td>
<td>-</td>
<td>1300</td>
<td>1100</td>
</tr>
</tbody>
</table>

### 4. EXPERIMENTAL RESULTS

Mild conditions can be practically identified by both the flame disappearing and the reduction of temperature gradients inside the furnace, that means a reduction of NOx emissions, as reported in figure 2 for a typical experimental run with methane, carried out with a SN configuration.

First of all, it has been studied the influence of the equivalence ratio (φ) of the reactants mixture, in the range 0.9-1.11, on mild combustion sustainability of methane. NOx and CO emissions have been monitored at given dilution ratios (Kv), then expressed on a dry basis and normalized to the same reactants flow rate. As evidenced by figure 3a, a reduction of the equivalence ratio from 1.11 to 0.9 did not significant affect NOx emissions profiles in mild conditions, even if the dilution ratio Kv is increased by an external N2 addition (Kv=6.9).
Theoretically, a larger oxygen excess should stabilize traditional flame conditions, reducing the possibility to sustain mild combustion; this was not observed due to the strong effect produced by the internal dilution, caused by exhausts entrainment within the reactants jet, that rapidly creates a low oxygen environment suitable for mild combustion conditions. Anyway, a slightly different effect was evidenced for CO emissions, as highlighted in figure 3b. An increase of the dilution ratio, in the whole range of equivalence ratios investigated, reduces the complete fuel oxidation, increasing CO emissions up to 10-15 ppm.

In the SN configuration, as previously mentioned, a partial premixing between air and fuel occurs before they exit the nozzle. This means that fuels can start the oxidation reaction before they reach the region of the maximum Kv, increasing the flame stability and producing a larger pollutants amount. Therefore, a switch of the reactants feed from a SN to a DN configuration was evaluated; mild conditions have been obtained with a SN configuration and subsequently, the fuel flow rate was switched progressively towards a DN configuration. No CO emissions have been practically found during the tests and the average furnace temperature was almost constant at about 980°C. NOx emissions, as shown in figure 4, were slightly increased when methane is fed from both the nozzles, while they reached a low concentration level when the fuel is completely fed from the secondary nozzle. It has been demonstrated that in mild conditions the system has a strong stability and that it is possible, also in a small-scale apparatus, to feed separately fuel and air without perturbing mild conditions.

Fig. 2  Typical experimental profiles for CH4 as fuel: (a) temperature and (b) emissions.

Fig. 3  Influence of the equivalence ratio (ϕ) on (a) NOx and (b) CO emissions at different dilution ratios (Kv); CH4 as fuel, SN layout.

Fig. 4  NOx as a function of the amount of CH4 fed through the 2nd nozzle (O2 excess = 3%).
5. NUMERICAL RESULTS

CFD results may help assessing the internal flow pattern during the burner operations. As evidenced by the streamlines depicted in figure 5, where the flow fields for different cases, concerning flame for the SN layout (run 1), mild combustion for both SN (run 2) and DN layout (run 4) are reported, the recirculation loop is well recognizable. It can be observed that the injection of the reactants jet entrains combustion products promoting their recirculation. In addition it can be noticed the presence of a dead region in the lower part of the burner, at least for run 2 where the secondary air co-flow was not used. The dilution ratio into the reaction region can be computed, by using simulation results, through the following relationship:

\[ k_v = \frac{\int \rho_E v_E dA}{m_f} \tag{1} \]

where the area \( A \) is perpendicular to the burner axis, \( \rho_E \) and \( v_E \) are the exhaust density and negative velocity, respectively, while \( m_f \) is the fresh reactants flow rate fed to the burner. These results are summarized in figure 6, where the computed \( K_v \) were plotted vs. the non-dimensional distance from the nozzle tip (\( z/D, D=\)nozzle ID=3 mm).

![Fig. 5 Velocity streamlines (m/s) representing different burner operations with CH4 as fuel.](image)

![Fig. 6 \( K_v \) values induced by the jets vs. the non-dimensional distance from the nozzle.](image)

Figure 6 clearly point out the influence of the large recirculation loops shown in figure 5; indeed, in mild combustion conditions the dilution ratio depends only from the high velocity jet of combustion air, also in the case of a lateral injection of CH4 (run 4). On the other hand, a low dilution was found for the flame condition (run 1), characterized by low velocities in the axial jet and a massive air co-flow outside the nozzle. This behavior dumps the recirculation of exhaust gases into the reaction region and, as a consequence the possibility to reach stable mild conditions; figure 7 reports the \( O_2 \) molar fraction contours within the
combustion chamber for different cases: runs 1 and 3, in flame regime, where strong O\textsubscript{2} gradients can be found especially around the flame, and run 5 where the oxygen is strongly diluted inside the chamber before to reach the fuel jet. The products distribution is not symmetrical; for the DN layout, this behavior is due to the fact that CH\textsubscript{4} enters the combustion chamber from a lateral port, while for the SN configuration can be ascribed to the aforementioned partly premixing of fuel and combustion air in the upper part of the nozzle, as shown in figure 8, where several contours evaluated at the nozzle tip are reported for run 2.

![Fig. 7](image)

**Fig. 7**  O\textsubscript{2} molar fraction computed for different burner operations of table 2.

![Fig. 8](image)

**Fig. 8**  Example of several contours obtained in single nozzle simulations (run 2 of table 1).

6. REFERENCES