MODELLING AND FULL-SCALE TESTING OF A 35 MW HEAVY-DUTY MILD BURNER

D. Ettorre*, A. Puzo*, T. Vela, S.B. Ahmadpanah**, A. Saponaro***, M. Torresi*, and G. Rossiello****
corresponding author gianluca.rossiello@seamthesis.com
* DMMM, Department of Mechanics, Mathematics and Management, Polytechnic University of Bari, Via Re David, 200 – 70125 Bari, ITALY
** SEAMTHESIS Srl, Via IV Novembre, 156 – 29122 Piacenza, ITALY
*** Centro Combustione Ambiente SpA (CCA, Sofinter group), via Milano km 1,600, 70023 Gioia del Colle (BA), ITALY

Abstract

This research is focused on the experimental testing of a Ultra Low-NOx burner for steam generators, at its full capacity of 35 MW of fuel thermal input, and on the CFD modeling and simulation in three operating conditions, with comparison of NOx emissions coming from numerical results against experimental data.

First, the burner and the experimental test-rig, represented by a 48 MW boiler by the CCA facilities in Italy, are shown and illustrated, along with the testing conditions and the results obtained in terms of performance and emissions, in three different operating conditions. In all three cases the burner is fed with natural gas, the first being the baseline. In the second test condition steam is injected to reduce the NOx production, while in the third one EFGR (External Flue Gas Recirculation) is employed, keeping constant the steam mass flow rate, thus accomplishing the 1-digit target on the nitrogen oxides emissions.

In the second part CFD simulations are shown to give much insight in the burner operation and flame behavior inside the furnace, and to compare the overall results obtained in terms of performance, *i.e.*, NOx emissions predicted. The NOx modeling is carried out on the computed combustion temperature and species fields, and the outlet levels compared to the experiments. The results obtained prove that the chosen modeling is enough accurate to represent properly the main features of the complex physical phenomena involved, and the NOx estimation is found to be quite accurate to employ extensively the CFD tools for a detailed design process.

Context and motivation

Energy production and process industries account for more than one-third of global primary energy demand. The main source of this energy is still fossil fuel combustion. Since NOx is one of the most harmful emissions from combustion, the development of technologies that reduce its emission is of primary importance for industrial combustion applications. Combustion technology greatly influences NOx formation, and its development is the first essential step in limiting emission levels.

To strongly decrease the NOx emissions, new generation of Low-NOx burners (LNBs) is being developed [1]. The LNBs technology includes strategies such as air/fuel staging and flue gas recirculation, not necessarily as an alternative but often in synergy [2]. A particular mention deserves the MILD (Moderate or Intense Low Oxygen Dilution) combustion technology [3-5] where a dilution of the comburent air together with an increase of its temperature above the fuel self-ignition temperatures allows the delocalization of the oxidation reaction avoiding temperature peaks and NOx formation.

The key themes of this work are the analysis of the results of full-scale experimental tests on an industrial burner conducted on a dedicated test rig, and a comparison with CFD (Computational Fluid Dynamics) fluid dynamic simulations performed on the burner and combustion chamber assembly. Moreover, the research aims at identifying the potential of the MILD combustion applied to large scale industrial burners and to assess the effectiveness of specific techniques such as EFGR and steam injection for NOx emission reduction, also showing how NOx emission evaluation by CFD is suitable for very large-scale test equipment and compares well even with low NOx levels under extreme MILD combustion conditions.

Problem description and experimental results

CCA has developed a new Ultra-low NOx burner, in the 35 and 45 MW scale, which have been studied and optimized by means of CFD analysis and tested in full scale at the CCA facility in Gioia del Colle (Italy).

The test rig, in which the experimental tests performed on the burner are carried out, of the Center for Combustion Environment (CCA) (Figure 1), is equipped with a test combustion chamber for burners up to 48 MW thermal size, including feeding systems (gas, liquid and solid pulverulent), flue gas cooling (water-steam circuit, with evaporator tubes, superheater banks and condenser), flue gas treatment system with bag filters, and flue gas expulsion.

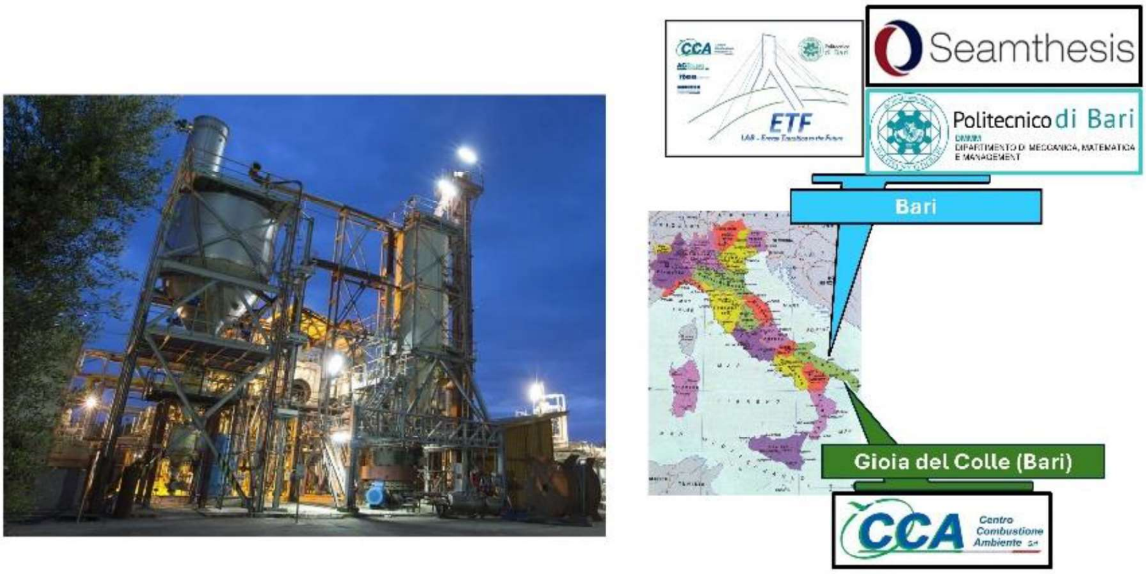

Figure 1. CCA 48 MW test-rig in Gioia del Colle (right) and 35 MW burner (left).

These tests are conducted by keeping constant the thermal input of 35 MW thermal referred to the lower heating value of the fuel, i.e., natural gas. This corresponds to 100% of the nominal load and the air flow rate is made to vary to test different excess air values, as well as the steam flow rate and the recirculation rate of external flue gas to evaluate its effect on NOx reduction.

During the tests, the quantities of interest are measured and recorded, Figure 2 shows the history of a test day, where NOx and CO are shown in ppm on a dry basis and normalized to a flue gas oxygen content of 3 percent on a dry basis. Also shown in Figure 2 is the dry oxygen content in flue gas (in percent) and the EFGR flow rate (in t/h). Six subdivisions are visible in the graph, representing the time intervals over which the results are averaged to identify significant operating points. Once the experimental testing has taken place, three main operating conditions of interest are identified: the baseline in period A (neither combustion gas recirculation nor steam injection is present), a second with steam injection in interval B, and a third with combined steam and combustion gas recirculation action in interval E.

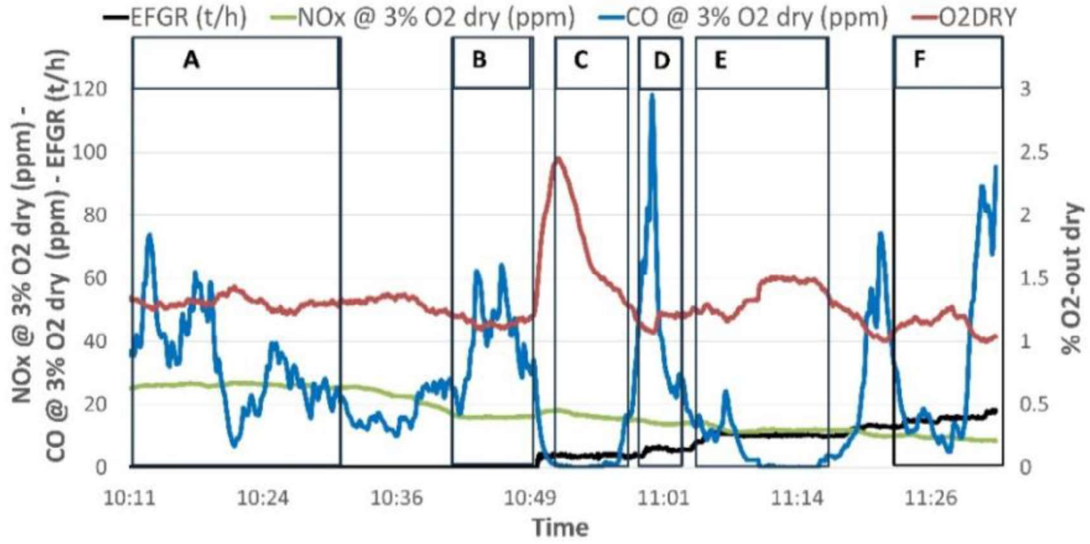

Figure 2. Test history chart (EFGR, O_2, NOx, CO) and operating points identified.

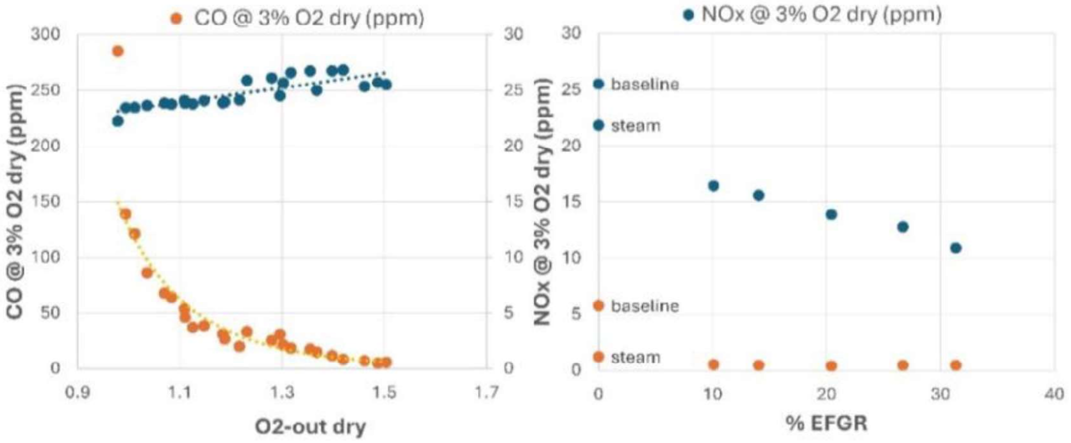

Figure 3. NOx and CO emissions depending on: (right) O_2 content in flue gases (*i.e.*, air excess), and (left) External Flue Gas Recirculation rate (%).

Figure 3 shows on the left the trend of CO and NOx production as a function of oxygen content obtained as a result of the analysis performed on the flue gases.
On the right are shown the trends of reaction products (CO and NOx) as a function of the percentage of recirculation of the flue gases obtained during the test. It is clear from those trends that the presence of EFGR, with low residual oxygen contents in the flue gas, generates a reduction in emissions in terms of NOx and CO production.

Numerical modeling and results

The quantities of interest associated with each case considered are shown in Table 1 and used to define the inputs for CFD setup and to perform the comparison of numerical simulation results against experimental testing. The fuel flow rate, the flow rate of combustion air input, are exploited. Emission values, in particular, are used to verify and validate the results of the experimental analyses.

Table 1. Magnitudes of interest for the cases considered.

Grandezza	**Units**	**Case 1: BASELINE**	**Case 2: STEAM**	**Case 3: STEAM + EFGR**
MFR air	*Kg/s*	*13.345*	*13.246*	*13.449*
MFR fuel	*Kg/s*	*0.79*	*0.79*	*0.79*
MFR H2O vapour	*Kg/s*	*0*	*0.239*	*0.239*
MFR EFGR	*Kg/s*	*0*	*0*	*2.869*
O_2 *dry*	*%*	*1.3*	*1.14*	*1.48*
NOx dry, 3@O_2	*ppm*	*26*	*16*	*12*
CO dry, 3@O_2	*ppm*	*50*	*50*	*0*

CFD analyses are conducted with the Star-CCM+ simulation suite [6]. The computational domain, depicted in Figure 4 on the left, is discretized with a polyhedral mesh of about 5 million elements with orthogonal prismatic cells next to the walls and generating smaller cells close to complex geometries (Figure 4 on the right), like the swirler and the gas spud, to correctly capture flows and combustion details.

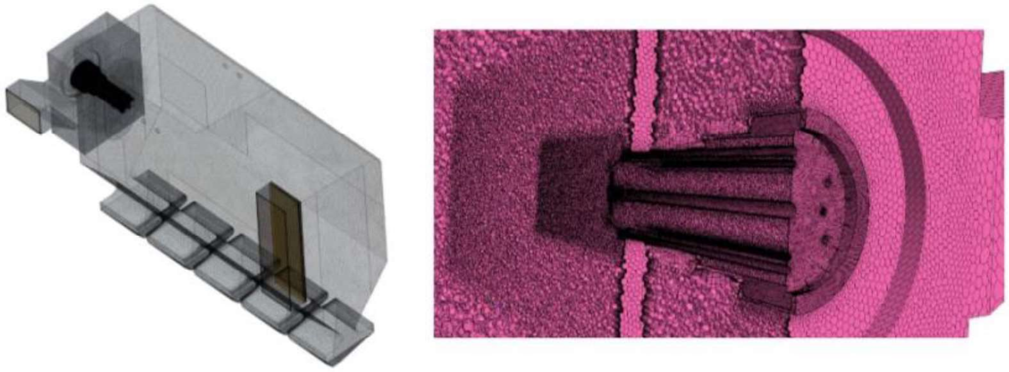

Figure 4. View of the mesh domain (left) and detail in the burner area (right).

For each configuration, a steady state computation is performed with turbulence modeled by a RANS approach and standard 2 equations closure. A simplified chemistry approach has been employed, and the combustion is treated with the Flamelet Generated Manifold (FGM) [7]. The 325 reaction and 53 species mechanism GRI-Mech 3.0 [8] is chosen to model both the natural gas combustion and the NOx emission production. Compressible ideal gas approach is employed for the equation of state and to account for radiation a Discrete Ordinates Method is used, with absorption coefficients calculation based on Weighted Sum of Grey Gases Model, in which a mean optical path is computed on the whole fluid domain.

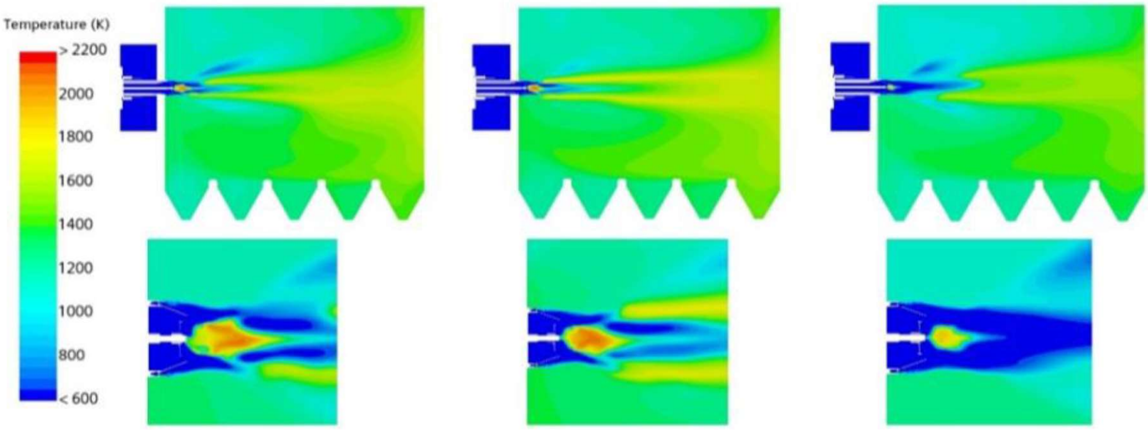

Figure 5. Contours of temperature in the XY plane (z=0), Case 1 on the left, Case 2 middle, Case 3 on the right.

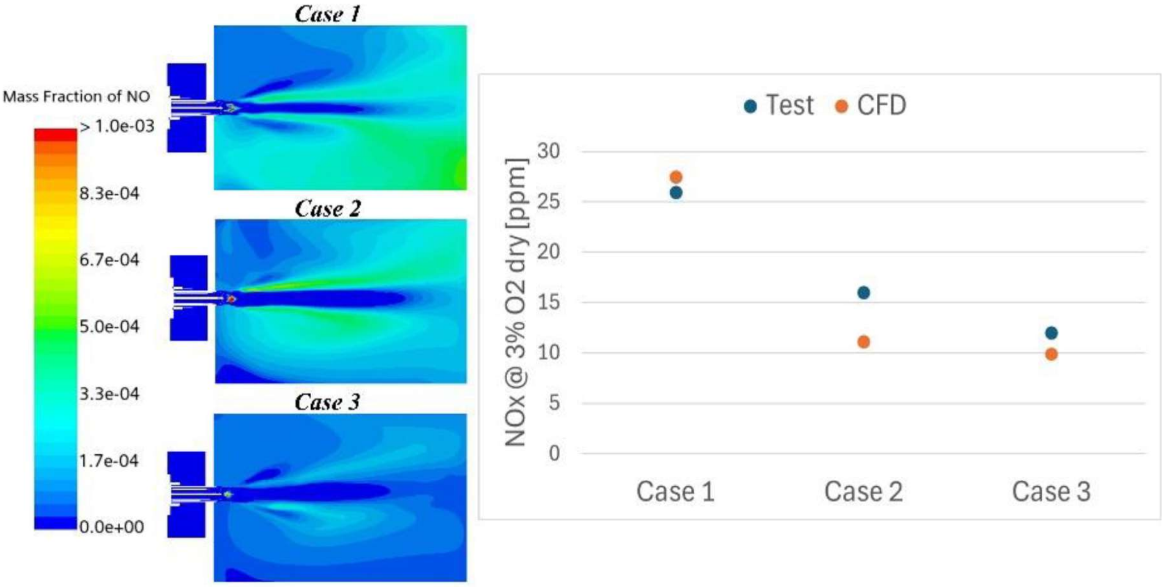

Figure 6. Contours of the NOx mass fraction on the XY plane (z=0) (left) and comparison between the NOx emissions on a dry basis, 3@O2 measured in the tests and the numerical values (right).

From the study carried out, the three cases analyzed exhibit a relatively uniform temperature field (Figure 5), with moderate local temperature peaks, a sign of the recirculation of combustion gases occurring within the chamber, typical condition for the MILD-type combustion case. However, higher gas temperatures characterize the burner near field, where a pilot stabilization flame is used (see Figure 5, pictures in the bottom row, with zoom of burner near field).

As shown in Table 1, the steam injection configuration resulted in a reduction of NOx of 35% compared to the standard operation, and the addition of EFGR, causes a further decrease of NOx is measured, with a total reduction of 50% opposed to Case 1. This behavior is also predicted well by the CFD simulations, as can be seen in Figure 6 from the NOx mass fraction contours (on the left) and the comparison of the results obtained from the experimental data and simulations (on the right), which demonstrates good concordance on the values and trend of NOx.

Conclusion

In this work, an industrial Low-NOx burner is experimentally and numerically investigated in three different operating conditions. The investigation leads to the convergence of experimental tests and computational approaches, enabling an in-depth understanding of the phenomena governing MILD burner behavior, thus through the integration of experimental data and CFD simulations.

The test data allows validation of the adopted numerical model, and the considerable agreement between CFD results and experimental values of the NOx emissions shows that this methodology is reliable and usable for the design of industrial equipment, from the early stages up to the optimization of geometries.

References

[1] J.M. Ballester, C. Dopazo, N. Fueyo, M. Hernández, P.J. Vidal, “Investigation of low-NOx strategies for natural gas combustion”, *Fuel, 76, 1997*, 435-446.

[2] E.C. Zabetta, M. Hupa, K. Saviharju, “Reducing NOx emissions using fuel staging, air staging, and selective noncatalytic reduction in synergy”, *Ind. Eng. Chem. Res., 44, 2005*, 4552-4561.

[3] M. De Joannon, G. Langella, F. Beretta, A. Cavaliere, C. Noviello, "Mild Combustion: Process Features and Technological Constrains”, *Combustion Science and Technology, 153, 2000*, 33-50.

[4] J.A. Wünning, J.G. Wünning, “Flameless Oxidation to reduce thermal NO-formation”, *Progress in Energy and Combustion Science, 23, 1997*, 81-94.

[5] I.B. Özdemir, N. Peters, “Characteristics of the reaction zone in a combustor operating at mild combustion”, *Experiments in Fluids, 30, 2001*, 683-695.

[6] Simcenter STAR-CCM+ 2023.06 User Guide.

[7] Van Oijen, J. A. and De Goey, L.P.H. Modelling of premixed laminar flames using flamelet-generated manifolds. Combust. Sci. Technol., 16:113, 2000.

[8] http://www.me.berkeley.edu/gri_mech/