Available online at www.sciencedirect.com

ScienceDirect

Energy Procedia 81 (2015) 967 – 976

Energy Procedia

69th Conference of the Italian Thermal Engineering Association, ATI 2014

CFD analysis of NOx emissions of a natural gas lean premixed burner for heavy duty gas turbine

A. Andreini[a], M. Cerutti[b], B. Facchini[a], A. Innocenti[a,*]

[a]*Department of Industrial Engineering of Florence, University of Florence, Via di S. Marta 3,Florence 50139, Italy*
[b]*GE Oil & Gas, Via F. Matteucci 2, Florence 50139, Italy.*

Abstract

The present work presents a numerical analysis of a low NOx partially premixed burner for heavy duty gas turbine. The first part of the paper is focused on the study of the premixing process inside the burner using standard RANS CFD approach. The resulting profiles at different test points have been used to perform reactive simulations of an experimental test rig, where exhaust NOx emissions were measured. A reliable numerical setup was found comparing predicted and measured NOx emissions at different operating conditions and split ratios between main and pilot fuel. The calibrated numerical setup was then employed to explore possible modifications to fuel injection criteria and fuel split, with the aim of minimizing exhaust NOx emissions. This preliminary numerical screening of new fuel injection strategies, allowed defining a set of advanced configurations to be investigated in future experimental tests.

Peer-review under responsibility of the Scientific Committee of ATI 2014

Keywords: CFD; NOx; Combustion model; Premixer design

---

1. Introduction

Lean premixed combustion technology can be considered the most effective solution to meet the stringent regulations on pollutant emissions, in particular NOx, of these last years. The fuel and air are premixed within the injector, to avoid the formation of non-uniform near-stoichiometric local mixture composition inside the combustor, thus allowing a more precise control on flame temperature. With the aim of obtaining uniform mixing between fuel and air, cross-flow jet (JCF) configuration is widely used to enhance mixing in premixers injection systems. The jet mixing that occurs in the premixing system becomes thus of prime importance and plays a fundamental role in achieving satisfactory combustion performance.

* Alessandro Innocenti . Tel. +39 055 2758771
E-mail adress: alessandro.innocenti@htc.de.unifi.it

Peer-review under responsibility of the Scientific Committee of ATI 2014
doi:10.1016/j.egypro.2015.12.155

Nomenclature

V	Scaled velocity	[-]
k	Turbulence kinetic energy	[-]
ϕ	Equivalence ratio	[-]

Subscripts

t	Turbulent
ref	Reference

As the flame gets close to its extinction limits, stabilisation problems arise due to the occurrence of thermoacoustic instabilities. One of the methods adopted to stabilise the flame in real engine combustion chamber is to combine premixed combustion with small diffusion flames (pilot flames) sustaining combustion process, preheating the reacting mixture.

CFD calculations are now truly competitive with experiment and theory, as a research tool to produce detailed information about combustion processes and play a crucial role in the design of environment-friendly devices. Despite more detailed approaches (i.e. Large Eddy Simulations) which are usually still out of the capabilities of the R&D departments of many companies involved in energy and power generation business (1), steady RANS calculations are still the main industrial numerical tool to investigate flame structure and its stabilisation dynamics. Moreover, when the main interest is in terms of trends, with respect to key design inputs, RANS predictions succeed in providing reliable results when applied to reactive cases and pollutant emissions.

In the present paper, a numerical investigation has been carried out, performing RANS simulations, to study the aerodynamics and the mixing evolution inside a single premixer where the fuel is injected in a JCF configuration. Obtained exit profiles were compared with archived experimental data from GE Oil&Gas past experience and given as boundary condition for the following reactive simulations. Exhaust emissions were then evaluated and compared with available experimental measurements on an experimental reactive test rig at different operating conditions and different split ratio between main and pilot fuel.

The assessed numerical setup was then employed to explore possible alternative fuel injection strategies and fuel splits getting down to NOx emissions reduction. Different injection patterns were tested aimed at obtaining more intense mixing inside the premixer and more uniform fuel concentration profiles, thanks to an enhanced fuel jet penetration and to the exploitation of different injection locations. This preliminary numerical screening allowed the definition of a set of advanced configurations.

2. Premixer internal aerodynamics

2.1. Premixer Geometry and Numerical Setup

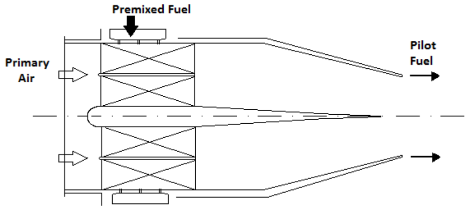

Fig. 1 DACRS premixer scheme

The investigated premixing system consists in a Dual Annular Counter-Rotating axial Swirl (DACRS) nozzle. A schematic cross section of the geometry is represented in Fig. 1.

The fuel is injected in a JCF configuration from the outer annulus by feeding holes. A converging nozzle, where the mixing is completed, follows the swirler before entering the combustion chamber.

The premixer periodicity allows the simulation of only 1/5 of the whole geometry (see Fig. 2), thus reducing numerical costs of the simulation. Two separated sectors were created, each one following the corresponding swirler rotation, merging them in one sector at the converging nozzle inlet. In order to guarantee flow continuity, a fluid-fluid interface (General Grid Interface (2)) is introduced for top and bottom surfaces of respectively inner annulus and outer one.

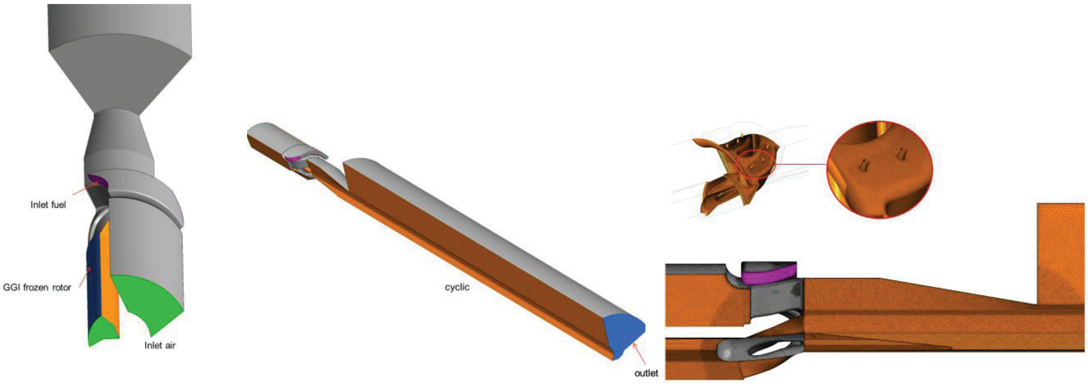

Fig. 2 Premixer geometry and mesh particulars

All the simulations were performed with ANSYS CFX 14.0 on a 7.2 E6 elements mesh in Fig. 2.

As far as the numerical setup is concerned, a specified blend factor scheme (2) with a blend factor of 0.5 has been adopted for momentum and CFX high resolution scheme for species transport equation. No slip condition was assigned on swirler walls, centre body and on the converging duct. Air and fuel were at the experimental conditions, that is, at 1 atm and at a 293 K. An opening boundary condition was instead assigned on the outlet section. Isothermal flow hypothesis was introduced.

After an assessment procedure, a constant Sc_t of 0.2 in scalar transport equation is chosen together with a tuned k-ε turbulence model, resulting from previous works on the topic (3). It is, in fact, well known that RANS models tend to underestimate turbulence levels when simulating non-stationary phenomena generated by the swirling flows or JCF configuration as well as high turbulence levels in the shear layers (4), (5), (6). In order to overcome the mentioned modelling difficulties, a modified model configuration was proposed where a value of 1.15 (instead of the default one of 1.44) has been proposed for the model constant $C_{\varepsilon 1}$ and a value of 0.2 for Sc_K in k equation instead of 0.9.

2.2. Results and Discussion

Obtained results are compared with experimental data at two reference sections of 1 mm and 32 mm far from the premixer outlet, at the end of the converging nozzle (see Fig. 3), in terms of Fuel Air Ratio (FAR) profiles.

Calculated FAR is normalised respect to a reference value FAR_{tot} and plotted against the radius R normalised respect to the external radius R_{ext}. R = 0 represents the domain axis. The experimental profiles are correctly represented at both the axial locations. CH_4 diffusion, from the injection point on the outer swirler to lower radii, is well predicted by the simulation with the proposed turbulence model.

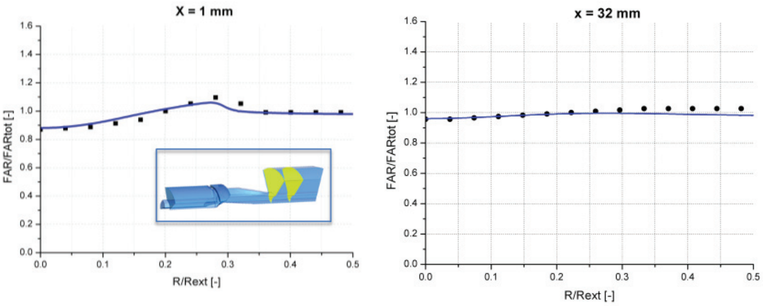

Fig. 3 Normalized FAR profiles obtained at two axial location from premixer exit

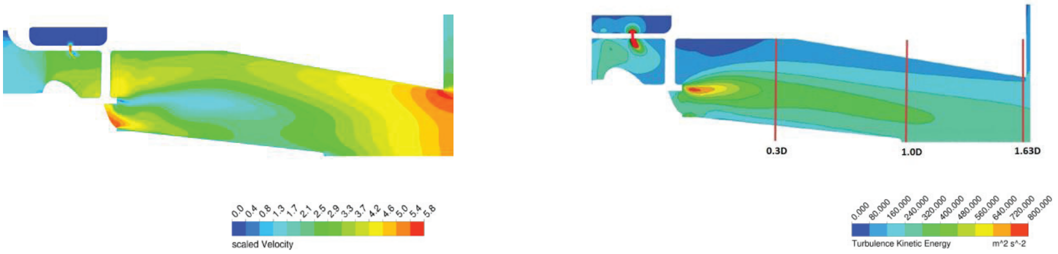

Fig. 4 Velocity (left) and turbulence kinetic energy (right) contours

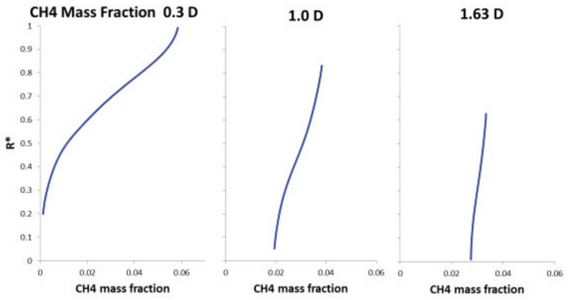

Fig. 5 Fuel mass fraction profiles at three axial locations

From velocity and Turbulence Kinetic Energy (TKE) contours in Fig. 4 the main features of the premixing system are evident, in particular, the JCF region and low-velocity region of the shear layer generated by the interaction of the two swirling flows. It is in this region that the highest levels of turbulence are observed. Despite the high turbulence level in the first part of the shear layer, from fuel evolution profile along the premixer in Fig. 5, is possible to see how CH_4 remains confined at high radii since 1.0D section where it has reached the premixer axis.

This first study gave important insight on the main aerodynamic features of the baseline burner as well as useful information about how the injected fuel diffuses along the premixer. Both these aspects are key points for the following phase of concept design of alternative injection strategies aimed at the optimisation of the mixing since the first section.

3. Reactive analysis

3.1. Test rig geometry and numerical set up

The investigated burner totally follows LPC concept design as it is based on a very lean premixed flame surrounded by discrete pilot injection points which help stabilise the flame. Reactive simulations of the experimental test rig with a single premixer, representing a single burner arrangement (see Fig. 6), have been performed.

The computational mesh resulted in 12.3 E6 cells (Fig. 6) with localised refinements at the injection system of pilot flames and a progressive coarsening towards the outlet. Mixture fraction, its variance, temperature, turbulence related quantities and velocity components profiles, to be assigned at the premixer inlet, are obtained after performing isothermal calculations, with the above described setup, at engine full load operating conditions.

After a turbulence model assessment, a standard k-ε model has been chosen, where the C_{e1} model constant was set to 1.30 to guarantee a sufficient TKE level inside the domain. Such a change helps limit intrinsic instability and reach convergence, thanks to the higher induced turbulent diffusion. For further details please refer to (3).

A partially premixed combustion model resolving both mixture fraction and reaction progress variable *c* was adopted. CH_4 without NOx opposed jet flamelet was generated from Peters' C2 detailed mechanism available in CFX, with 28 species and 100 reactions (2). Progress variable source term is modelled using Zimont's turbulent flame speed closure where an empirical laminar flame speed S_l correlation, based on opposed jet laminar flame results with GRI 3.0 mech has been used. A constant Sc_t of 0.5 and unity turbulent Lewis number were assumed.

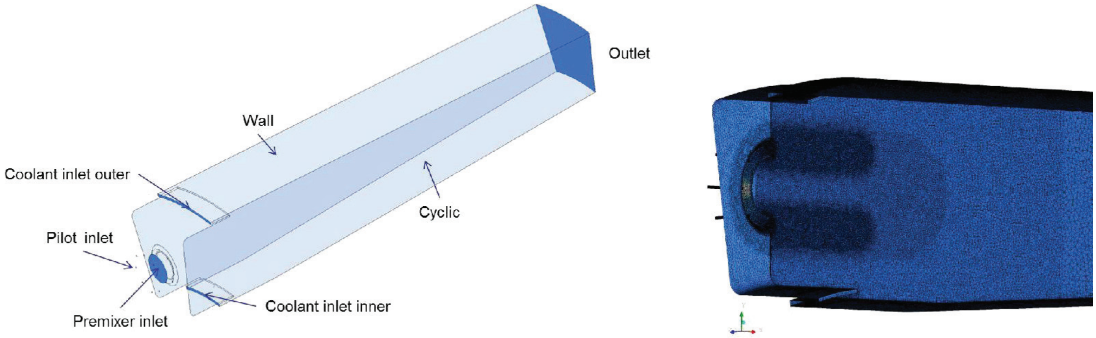

Fig. 6 Reactive test rig geometry and mesh

No slip adiabatic boundary condition was assigned at the combustor walls. Rotational periodicity was assigned on the two lateral surfaces. At pilot and premixer inlet a $c = 0$ boundary condition was assigned while $c = 1$ was set at the coolant inlet to let the process be governed by mixing only in such region.

NOx emissions were evaluated in a post-processing calculation, considering their influence on the flow-field, stable species and important radicals as well as on the heat release process negligible. An additional conservation equation for the mass fraction of NO is solved. Turbulence effects are taken into account averaging the laminar reaction rate with local β Probability Density Function (PDF) of a normalized temperature and of its variance. For further details refer to (7). Temperature fluctuations, necessary to calculate the PDF, are evaluated with a second additional conservation equation for temperature variance. Instantaneous laminar reaction rate has contributions from thermal and prompt formation mechanisms. The extended Zeldovich mechanism is considered to model the former. Radical O is calculated from O_2 under equilibrium assumption while for OH radical partial equilibrium is assumed (8). De Soete model for CH_4 (9) is used for Prompt NO. NOx emissions routines were implemented as an external model in the commercial CFD code ANSY CFX 14.0.

In the studied burner, NOx are expected to be essentially formed in high temperature zones near pilot diffusion flames, where Zeldovich mechanism prevails.

In order to obtain a complete map of NOx emissions at different premixer equivalence ratio ϕ and fuel spit s conditions, several test points have been simulated. Test Point 0 is chosen as the reference one (Table 1):

Table 1 Reactive analysis test matrix

Case Name	Fuel Split s [%]	ϕ/ϕ_{ref}
Point-0	15	1
Point-1	15	1.08
Point-2	15	0.84
Point-3	25	0.99
Point-4	5	1.01

3.2. Results and Discussion

The predicted flame shape and temperature are shown in Fig. 7. The premixed flame appears to be extended with a thick flame brush. The flame front is corrugated in its initial part due to the instabilities generated by the interaction with the pilot flames.

Normalised temperature distribution in Fig. 7 clearly shows the peaks due to the rich pilot flames. From vector plot in the same figure it is possible to observe cooling air entering the domain and being convected backward to the corner zone, interacting directly with the pilot flame thus leading to an unstable attachment of pilots and causing oscillations of the latter which is moved towards the premixed flow.

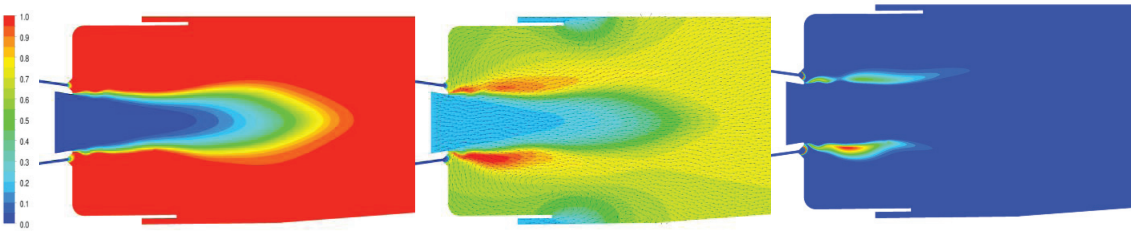

Fig. 7 Point-0 progress variable (left), mean temperature (centre) and NOx reaction rate (right) contours

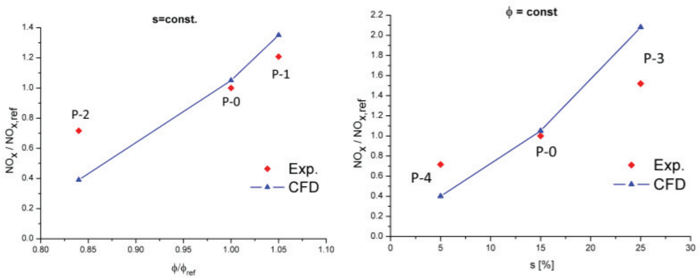

Fig. 8 Normalised NOx emissions at constant pilot split and constant equivalence ratio

As far as NOx emissions are concerned, in Fig. 7 it is possible to see that the diffusive pilot flames are actually the main responsible for NO formation, while premixed flame influence seems to be limited.

A good agreement was found in emissions trends prediction (Fig. 8). The relative variation rate, however, was not properly predicted by the model. The influence of the premixer on NOx formation rate was found to be marginal if compared with the pilot flame one (note the different scale in the two plots).

The underprediction of NOx emissions at Point-2 may be due to a not correct estimation of the laminar flame speed at such lean conditions, leading to some difficulties in quenching prediction and in the evaluation of its effects on NOx levels. A too high sensitivity to thermal formation mechanism is also emerged. When a very lean flame (Point 2) or a low *s* case is studied (Point 4), a reduction in temperature levels is predicted and consequently, thermal mechanism weight on the final prediction is reduced and NOx levels underpredicted. The opposite behavior is observed for Point 3.

4. Alternative injection configurations

4.1. Isothermal investigation

In the final part of the work two alternative injection system configurations (Fig. 9) have been proposed and tested:

- SW1 with enhanced jet penetration thanks to higher jet velocities.
- SW2 with a more uniform radial distribution of the injections.

SW1 SW2

Fig. 9 Alternative injection configurations: geometry and mesh

Numerical setting and turbulence model for the new isothermal simulations is the same employed for the baseline investigation. SW1 mesh is around 6.5 E6 elements while SW2 ones is around 6.8 E6 elements.

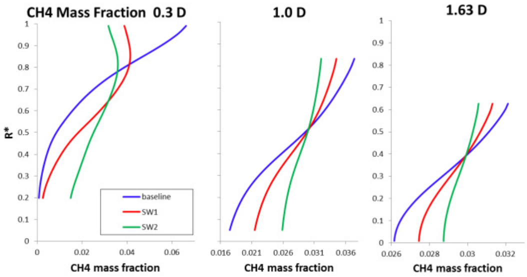

Fig. 10 Fuel mass fraction profiles at three axial locations for the two proposed geometries and baseline one

From 0.3 D CH_4 mass fraction plot in Fig. 10 it is possible to observe the effects of the different injection configuration on the mass fraction distribution at a short distance from the swirler exit. The baseline configuration presents a high fuel concentration at higher radii. The fuel does not sufficiently penetrate to have an as uniform distribution as for the other cases. At low radii the fuel concentration is increased in the two proposed configurations. In SW1, thanks to a deeper penetration of the jets (see Fig. 11), higher fuel concentrations are achieved at low radii since the first sampling plane (0.3D). The premixer outlet profile results to be more uniform.

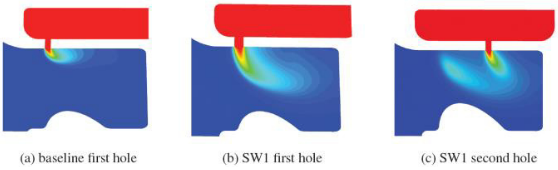
(a) baseline first hole (b) SW1 first hole (c) SW1 second hole

Fig. 11 Fuel penetration for baseline hole and the two holes in SW1

In SW2 configuration the fuel is injected, in addition to the outer swirler annulus, also on the outer swirler suction side and on the inner swirler annulus. The resulting fuel distribution along the radius can be appreciated from Fig. 10 at the considered axial locations. Since the first plane, SW2 geometry allows an optimal distribution, having the highest concentration towards the premixer axis.

4.2. Reactive analysis: results and discussion

Extracted profiles from previous mixing analysis on the alternative configurations have been given as input for the following reactive simulations. Geometry, mesh and numerical setting are maintained the same of baseline reactive analysis. Three reference test points have been chosen and tested to compare the alternative configurations in terms of NOx emissions: Point 0, Point 1 and Point 3. The effect of the new profile can be appreciated at two pilot split values and two ϕ. Moreover, a different pilot injection configuration (SH1) has been tested together with the baseline and the SW2 premixer injection configuration (which turned out to provide the lowest NOx emissions). The pilot ducts are modified changing their injection angle in order to provide a rotational component helping achieve more uniform mixture fraction and in turn temperature distribution.

The aim of this last part of the work is to provide a first insight on which design could lead to minimum NOx emissions in order to select configurations to be investigated in future experimental tests.

Table 2 Baseline and alternative geometries: NOx=NOxref [-]

Test Point	Baseline	SW1	SW2
0	1.05	0.96	0.86
3	2.08	1.97	1.84

Table 2 shows the results in terms of normalised NOx, with respect to the experimental value at test Point 0, obtained with SW1 and SW2 geometries at two test points. The improved uniformity at premixer outlet was found to have a direct impact on NOx emissions reduction. Results show that both SW1 and SW2 lead to a reduction of NOx ppm at the tested points and that SW2 design has a more marked effect. From Fig. 12 emerges that SW2 leads to lower emissions at all the tested points, independently of the pilot injection, split and ϕ.

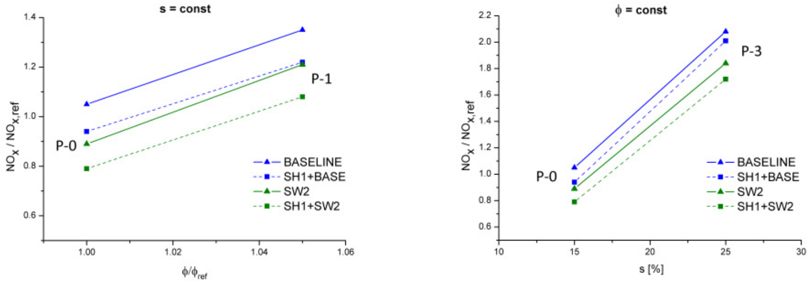

Fig. 12 NOx emissions predicted for the alternative designs

Results obtained with the modified pilot injection system are also reported in Fig. 12. From SH1+BASE results, it can be appreciated the only effect of the changed shroud. The SH1+SW2 configuration combines, instead, the features of both the designs.

As far as the alternative shroud is concerned, it seems that its impact on emissions is independent of the premixed fuel profile: the same improvement is obtained with baseline and SW2 premixer, for all the simulated conditions. From Fig. 12 it can be seen how a change from the original to the alternative shroud leads to a shift of the curves to lower NOx ppm, at both constant ϕ and s.

In all the conditions, the major contribution to NOx formation comes from the pilot flames. However, it is interesting to notice that the more uniform profile provided by SW2 premixer allows a reduction in NO emissions of the same weight of the SH1 pilot injection design. When the proposed changes are applied together, the beneficial effect is amplified. A global reduction of 25%, 20.5% and 15.6% is obtained with respect to the baseline configuration at, respectively, Point 0, Point 1 and Point 3.

5. Conclusions

A numerical analysis of a low NOx partially premixed burner for industrial gas turbine applications has been carried out with the main aim of finding a reliable description of the turbulent mixing inside the premixer as well as of the flame in the combustor. NOx emissions are eventually evaluated and compared with available experimental measurements at the combustor outlet.

In the first part the baseline premixer aerodynamic and fuel mixing along the premixer were studied. Useful information for the next phase of alternative fuel injection strategies concept design were provided by this analysis.

As far as reactive simulations are concerned, the adopted combustion model was able to deal with both the diffusive and premixed combustion modalities. A good agreement was, in fact, found in the prediction of NOx trends. The relative variation rate was instead not accurately caught due to a lack of accuracy in flame and quenching effects prediction at lean condition. The influence of the premixer in the NOx formation rate was found to be marginal if compared with the pilot flame one.

Deeper investigations with more detailed combustion models should be tested (for example flamelet generated manifolds) with the aim of improving the description of the turbulence-chemistry interaction in both the diffusive and premixed flame regions.

Two different injection system configurations of the premixer have been proposed and simulated. Some general criteria were successfully exploited to obtain more uniform fuel concentration profiles, due to a deeper jet penetration in the premixer and a different radial distribution of the injection points. An alternative fuel pilot injection design was also simulated introducing a rotational component in pilot injections, helping achieve more uniform temperature distribution.

The effect on NO emissions of the alternative design is evaluated. Even though the pilot contribution to NO formation is the major one, emerged that a more uniform fuel profile at the premixer have an equal weight on NO emissions reduction of the pilot injection system optimization.

Indication on the advantages or disadvantages on emissions obtainable with the proposed designs resulted to be precious for design improvements of partially premixed like burners and allowed the selection of a set of geometries to be investigated in future experimental tests.

References

1. Andreini A, Cerutti M, Facchini B, Mangani L. Modeling of turbulent combustion and radiative heat transfer in a object-oriented CFD code for gas turbine application. In Proceedings of ASME Turbo Expo 2008, GT2008-51117. 2008.
2. ANSYS. ANSYS CFX-Solver Modeling Guide. release 14.0 ed. .
3. Andrini A, Facchini F, Cerutti M, Innocenti A. Numerical Analysis of a Low NOx Partially Premixed Burner for Industrial Gas Turbine Applications. ENERGY PROCEDIA. 2014; 45(ISSN:1876-6102): p. 1382-1391.
4. Ivanova E, Noll B, Aigner M. A numerical study on the turbulent Schmidt numbers in a jet in cross-flow. In Proceedings of ASME Turbo Expo 2012, GT2012-69294. 2011.
5. Galeazzo FCC, Donnert G, Cardenas C, Sedlmaier J, Habisreuther P, Zarzalis N, et al. Computational modeling of turbulent mixing in a jet in crossflow. International Journal of Heat and Fluid Flow. 2013; 41: p. 55-65.
6. Goldberg U, Palaniswamy S, Batten P, and Gupta V. Variable turbulent Schmidt and Prandtl number modeling. Engineering Applications of Computational Fluid Mechanics. 2010; 4: p. 511-520.
7. Andreini A, Facchini B, Mangani L, Asti A, G. C, Modi R. NOx emissios reduction in an innovative industrial gas turbine combustor (GE10 machine): a numerical study of the benefits of a new pilot pilot system on flame structure and emissions. In Proceedings of ASME Turbo Expo 2005, GT2005-68364. .
8. ANSYS. ANSYS Fluent-Theory Guide. release 14.0 ed. 2011.
9. DeSoete GG. Overall reaction rates of NO and N2 formation from fuel nitrogen. In Proceedings of the 15th International Symposium on Combustion, Pittsburgh. 1975;: p. 1093-1102.