Proceedings of the International Gas Turbine Congress 2003 Tokyo
November 2-7, 2003

IGTC2003Tokyo TS-145

Numerical/Experimental Methodology for the Retrofitting of Combustion Chambers for Gas Turbines

Francesco MARTELLI[1], Giovanni RICCIO[1], Paolo ADAMI[1], Giancarlo BENELLI[2], Giordano TANZINI[2]

[1] Department of Energetic "Sergio Stecco"
The University of Florence
Via S. Marta, 3 – 50139 Firenze, ITALY
Phone: +39-055-4796239, FAX: +39-055-4796342, E-mail: g.riccio@ing.unifi.it
[2] ENEL Produzione – Ricerca, Via A. Pisano, 120 – 56122 Pisa

ABSTRACT

The evolution in the field of gas combustion chambers represents an area of considerable interest. From a scientific-engineering point of view this interest is justified by the growing need to contain the level of emissions within the increasingly restrictive limits imposed by national and international laws. The need to reduce the production of pollutants such as CO, UHC and NO_x, has actually modified the design of combustion chambers, which have switched from the traditional solution with diffusion flame to the recent configurations with premixed or partially premixed flames. Of great help in this context is the development of integrated design methodologies, capable of using both numerical and experimental tools of different complexity. The main aim is to improve the design process while keeping reduced costs and time to produce modified solutions. The methodology described combines simplified codes that have been developed for a first step and quick analysis of the widest possible range of design solutions. A reduced set of design modifications for improvement of combustor emissions is then further investigated by means of CFD numerical simulations. The proposed design is finally tested experimentally on the combustion test bench. The methodology was applied to the retrofit an old generation combustor used on energy production systems. The integrated use of numerical and experimental tools, in conjunction with the ENEL-Ricerca Team, allowed the successful modification of the combustion system. The final release, in fact, meets the reduced level of emissions while maintaining both the original operational and reliability performances.

NOMENCLATURE

CFD	Computational Fluid Dynamic
D	Liner Diameter
DLN	Dry-Low NOx
Dp/p	Total pressure drop across the combustor to the inlet total pressure
GT	Gas Turbine
L/D	Length/Diameter
STD	Standard-baseline
p	Pressure
Sn	Swirl number
T	Temperature
U	Axial speed component
UHC	Unburned Hydrocarbons
Vmod	Speed module
Vrad	Radial Speed Component
x/L	Axial Coordinate/Liner Length
ϕ	Equivalence ratio

INTRODUCTION

The growing importance of environmental issues, with special regard to the release of pollutants into the atmosphere, prompted the designers to develop innovative solutions for the combustion systems of the gas turbines (Benelli et al., 1996). Dry Low NOx type of gas combustor have come into industrial use after 1995. The transition from diffusion flame to premixed one has experienced a strong impact on the combustion systems and has produced a revision of methodologies and design criteria. The present paper focuses in particular on the numerical-experimental methodology, successfully applied in updating a combustion system characterized by high NO_x emissions which was widely used in gas turbines designed in the '80s and which became operational in the '90s.

Four solutions available to conform the original combustion systems to the new emission standards were considered.

1. The installation of advanced DLN combustors with broad modifications to the machine and to its control system: promising NO_x reduction, but high installation costs.
2. The installation of water or steam injection systems capable of reducing NO_x by 40-50%: availability of water, higher CO.
3. The research of optimized solutions, requiring "minor modifications" to the injection system and/or to the holes of the combustor liner: limited NOx reduction but cheep and adequate for the retrofit (Feitelberg et al., 2000);
4. The combination of the above-mentioned "minor modification" solution with the injection of inert. This choice actually represents a hybrid solution of great interest to the users of old generation gas turbines. As a matter of fact, this solution combines the emissions containment with a low impact on the machine and with moderate consumption of inert (water or steam).

Modern combustors, (DLN), offer significant advantages but use combustion condition and injection systems which are much more complex than the traditional ones and their design and set-up is unquestionably more onerous (McVey et al., 1992). The second solution, which is widely used, has limited costs but can lead to considerable losses of efficiency and to an increase in running expenses with a decrease in performance, which is sometimes too high. In these cases the user is obviously interested in finding new targeted solutions, like the last one among the ones proposed, which offer a more economical operation. Nevertheless in this case an accurate re-design of the combustion system is needed through a systematic study for the optimization of the mixing air-fuel-inert, which is needed to define the modifications of the combustor (Jacob T. et al., 1995).

The development of new solutions implies, however, making a

Manuscript Received on April 4, 2003

new effort to revise the common practice for designing the conventional combustors and requires new tools for experimental investigation and numerical simulation. This calls for a more complex design stage, also in view of the several phenomena and aspects connected with a combustion system which requires organizing the resources available according to an integrated methodology which tackles the problem designing the gas turbine combustors in a concise and targeted way.

Generally speaking, the development of efficient combustors with low NO_x emissions requires structures and combustion systems which allow the designer to keep the equivalence ratio and the residence time in the flame zone at sufficiently low values (NO_x containment) while, at the same time, acceptable flame stability has to be guaranteed as well. This problem is parti cularly felt at partial loads where inadequate residence times to complete the oxidation of CO are easily obtained.

Moreover, the premixed type flames are often prone to the risk of self-ignition, flash-back and stability problems. Conversely the solutions adopted considering a partially premixed type flame allows an easier control in terms of stability and satisfactory operation at partial load still allowing a good reduction of the emissions. From this point of view partial-premixing may be considered a valid compromise for the retrofit of old generation combustors.

The present paper briefly describes the activity carried out in the framework of the project for the improvement of the combustion systems for gas turbine like the Fiat TG50. The objective of the project was researching low cost solutions which would allow constant operation and produce low emissions. For this purpose the gas injector was re-designed with the objective of increasing the quantity of air and the air-fuel mixing in the primary combustion zone, thus lowering the temperature of the flame, both globally and locally. The practical solution studied for the limitation of the emissions, consists in replacing the swirler with an axial type one, with fuel injection in the blade to blade vanes in place of the purely diffused flame of the original model. The fluid dynamics of the new premixing duct is of great importance with regard to the air/fuel mixing process and consequently the levels of emissions and combustion stability. For this reason it is of great importance to define a geometry capable of assuring a good mixing and a steady flow, practically without any oscillatory phenomena.

The design methodology followed in the first stage used simplified model (semi-empiric correlations) 0-D, 1-D to perform numerical predictions and for a quick preliminary analysis of the most suitable solutions. Afterwards the aerodynamics of the systems was investigated by means of CFD 3-D Full Navier-Stokes simulations to evaluate the correct aerodynamic behavior of the new components. The experimental contribution to the methodology was mainly dedicated to combustion tests on complete combustors (full size scale) both at atmospheric pressure and full pressure. The tests with atmospheric pressure were used in the preliminary stage for the selection of the most promising modified configurations. Accordingly it was possible to reduce considerably the full pressure tests to provide a last validation of the results achieved.

THE BASELINE COMBUSTOR

The combustor under investigation is an old generation model of the tubular “reverse-flow multican” type with diffusion flame. It is designed for heavy-duty machines with net electric power of about 120 MWe, Figure 1. The combustor equips the FIAT TG50D5 gas turbine machines, derived from the 701D model produced by Westinghouse-Mitsubishi, with an operating pressure of 1.36 MPa.

The dual fuel type injector is placed within the cylindrical duct of the dome and in axis with the liner. The fuel nozzle is placed on the central axis.

This paper considers only the operation with natural gas, which in the original model is injected directly in the primary zone by means of 12 nozzles. Upstream to the injector there is the annular duct for the swirl air which flows into the combustion chamber through the swirler, made of an array of 20 blades inclined with a 60° angle.

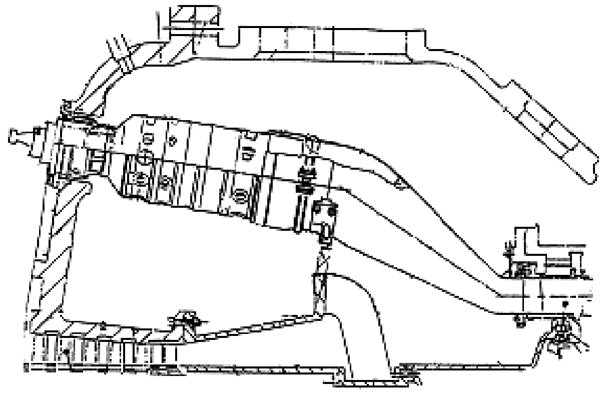


Fig. 1 The combustor placed inside the casing

The liner is made of a cylindrical shell consisting of a cylinder-conical shaped dome and of six ferrules with a wigglestrip type cooling system. The liner holes are composed of three arrays of 6 holes each, placed on progressive sections in the axial direction. All the holes are equipped with a scoop that goes inside the combustion chamber by about $0.1D_h$. The NO_x emissions control is accomplished by injecting demineralized water fed directly through the gas feeding pipe; the mixture fuel-inert flows into the combustion chamber through the nozzles placed on the head of the injector. The rate of water flow depends on the thermal load, thus the NOx emissions are kept within the 50 ppm@ 15% O2 limit with 100% load.

The use of water brings an increase in costs for four reasons: efficiency reduction, even though partially balanced by a stronger power produced; significant increase in the loss of sensible heat, request for production of demineralized water, more rapid deterioration of the machine with effects on the rate of maintenance intervals. From the operational point of view the injection of an excessive amount of water can determine considerable production of CO emissions thus limiting the operational range at partial loads.

The diagram in Figure 11 shows the curves of NO_x and CO emissions for the baseline version of the combustor, tested on the full pressure test-rig, fed by natural gas both for dry operation and with water injection. These values were taken as reference for the assessment of the results.

HYPOTHESIS OF MODIFICATION AND PRELIMINARY TEST

Two of the main factors controlling the emissions in gas turbine combustion chambers can be summarized as follows:

- Temperature and equivalence ratio in the reaction area, closely connected to air-fuel mixing “quality”
- Residence time of the combustion products in the reaction areas

The drawing of the modifications was made taking into account the geometrical availability allowed by a “minor modification” intervention on the combustor. At this stage semi-analytic models and correlations were used in order to have a general profile of the operation of the new items and of the combustor equipped with them.

The modifications refer to the operating conditions with dry natural gas, and set the objective of reducing the mean temperature of the flame by reducing the mean and local equivalence ratio of the primary area. This is obtained by increasing the air flow entering the primary area and taking it off the flow of the dilution holes; this solution also allows, while keeping the liner diameter unchanged, a reduction of the combustion products’ seat time, since the mean velocity of gases crossing the primary and secondary zone has increased. However, the reduction of the residence time must be such as not to quench the combustion reactions which would imply a notable increases in CO and UHC emissions. The uniformity of the local equivalence ratio values: ϕ in the primary zone can be obtained by the improvement of the air/fuel mixing process. For this is very important to study the air-fuel jet aerodynamic interaction, and consequently, to design the injection fuel holes position and diameter (Riccio et al., 2002).

These design guidelines resulted in the realization of a new injection system equipped with axial swirler with fuel injection holes placed directly inside the blading spaces, Figure 2. The air flow proceeds from the casing through the crossing sectors in the radial direction, then is diverted into the axial direction to cross the swirler, meet the gas jets and flow into the primary zone. The air flow rate is approximately double with respect to the original model. The modifications to the burner-swirler set aim at promoting a better air/fuel mixing (partial pre-mixing), in order to reduce the high temperature spots in the reaction zone. A first series of different models of these injection systems was tested on the combustion test bench at atmospheric pressure in order to evaluate, first of all, the actual working capability in terms of ignition performance, lean and rich blow out and operating range. Then the influence of the main design parameters, such as air flow rate, swirler angle, swirl number (Sn) has been investigated. The results of the survey on the combustor equipped with the modified injection system gave encouraging results in terms of lowering NO_x emissions (-30%) and defined a reference configuration. On the other side an increase in the maximum metal temperatures of the combustor in the dome area of about 40% was reported.

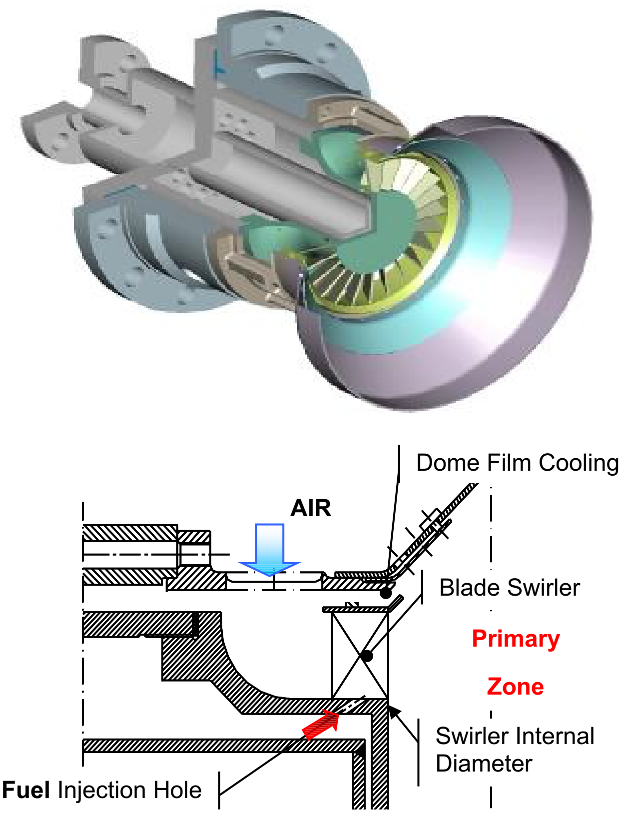


Fig. 2 Innovative injector for TG50D5 diffusion flame combustor and its longitudinal section

The most promising configuration has fuel injection from 10 holes drilled at the hub of the swirler in the blade-to-blade vanes. There was a decrease in the NO_x emissions reported for several intermediate models, as the air flow rate increases through the swirler up to a threshold value of about double with respect to baseline. Further increase did not seem to bring any advantage. The swirl number Sn was been kept rather low. In order to reduce the high metal temperature measured on the dome wall, a film cooling flow was introduced from the tip leakage of the swirler, Figure 2.

AERODYNAMIC STUDY OF THE COMBUSTOR BY MEANS OF 3-D NUMERICAL SIMULATIONS

The numerical simulation activity was included starting from the aerodynamic design and study of the preliminary configuration of the modified combustor defined in the previous phase.

In a preliminary phase 1-D model were employed to design the injection system. The aerodynamic of the swirler and the air fuel cross-flow were simulated by the 1-D model. Particularly the position and size of the injection fuel holes were designed considering the jet maximum penetration by means of the Lefebvre (1999) and Holdeman (1972) model implemented into a routine for the 1-D compressible flow calculation.

The objectives of the 3-D numerical simulations can be summarized in two main issues:

- Study of the injection system-swirler aerodynamics; check for possible operating problems such as, for example, region of flow separation.
- Study of the overall combustor aerodynamics, investigation of the new injection system with the complete combustor interaction.

The numerical simulations were organized in two different complementary groups in order to comprehend the phenomena under investigation:

- simulations dedicated to the study of the injector group
- study of the liner complete with all the inputs of air (primary, secondary, dilution and cooling devices holes) in order to evaluate the “swirler”-combustor interaction.

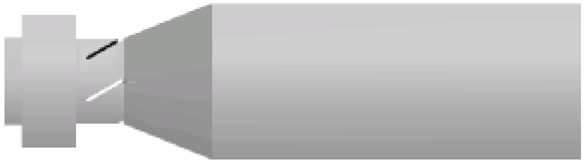
The first set of simulations makes it possible to study the characteristics of the injection system in details, allowing the investigation of the operation of the swirler and of the possible stall conditions that can take place on the blades. Then the simulations of the whole combustor aerodynamics allow mainly the assessment of the interaction between the modified injection system and the liner, thus verifying the effects on the aerodynamics of the complete combustor.

The performances in terms of emissions, pattern factor, temperature of the metal, operating field (lean & rich blow out), stability of the flame and emissions were verified mainly on a combustion test bench at atmospheric pressure and at full pressure.

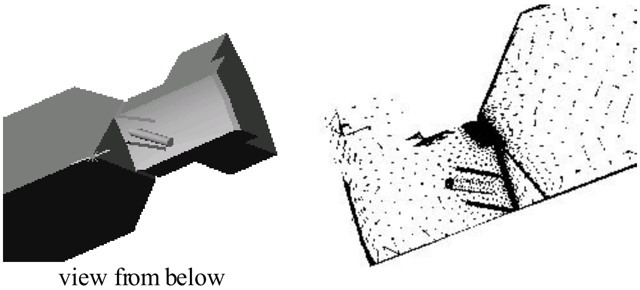
The solver used for the fluid dynamics computation is the Hybflow code, a new generation solver for internal and external domains. Hybflow is a 3-D code with finite hybrid type volumes autonomously developed by the Department of Energetics of the University of Florence (Adami et al., 2000, 1999) . The calculus grids, of the hybrid non-structured type, have been generated with a commercial tool developed by CENTAURSOFT^{TM}. Special care was taken in clustering the superficial mesh in the intersection area of the different channels representing the holes, and in the domain region characterized by large curvature. The grids are made up of about 500.000 elements. The calculation were carried out with turbulence model “Standard k-ω” and by imposing a ratio Dp/p of 0,05. The computation were performed on a Dec Alpha 500A CPU.

Results of the simulations of the modified injection system

The results of the aerodynamic study of the modified injection system are reported here below. The configuration taken into consideration is the one selected in the preliminary experimentation phase, at atmospheric pressure. The solid models, Figure 3, consider the geometry of the swirler in detail; this discharge in a volume with the same geometry of the combustor liner, but does not have the holes and the film cooling slot. The input geometry was realized in such a way so as to create an upstream area with respect to the swirler. Here a slow flow moves radially to mimic what happens in the real combustor. The CFD grid was realized by refining the swirler zone and the region immediately downstream, in order to increase the accuracy of the calculation in those regions of major interest.

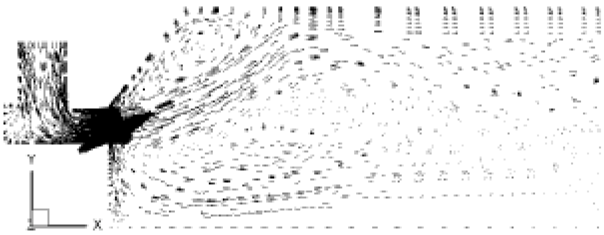


view from above

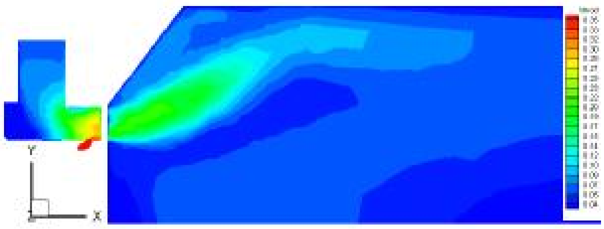


view from below

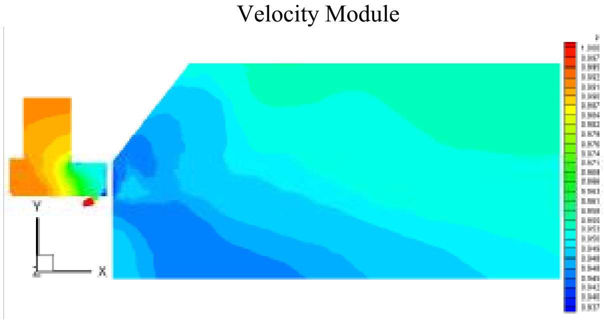
Fig. 3 Solid model and computation grid



Velocity Vectors



Velocity Module

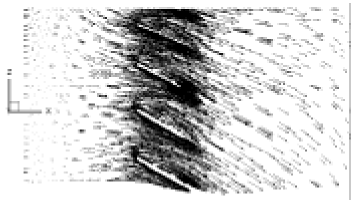


Static pressure

Fig. 4 Results of the simulation on a longitudinal plane from the axis of the model

Considering the results on a longitudinal plane, Figure 4, the formation of two main toroidal blow-by areas can be noted in the primary region. The more extended one is placed at the center of the discharge volume and has a considerable extension in the axial direction. The other one is placed in the area next to the conical wall, representing the dome. The two regions are separated by the flow coming from the swirler which, in the region of the dome, has a divergent main direction defined by the dimensions of the two main vortexes.

If we analyze the swirler in detail we note that the flow coming from the air box in the radial direction, undergoes an acceleration while approaching the axis of the model caused by the convergent geometry of the circular sector. Then the flow is sharply deviated by 90° and approaches the inlet section of the swirler. From the pressure maps it can be noted that the upper wall of the axial tract of duct, immediately downstream of the curve, shows a lower pressure region. In the configuration considered this determines a very small stall region, followed by the re-attachment of the stream, thanks to the blockage effect of the swirler. In the cases studied with a higher swirler cross section this phenomena was more evident and the separation on the upper wall extended towards the inner part of the swirler channel up to its exhaust. If we consider the results read on transversal planes downstream from the swirler, we can notice a good homogenization of the flow in the circumferential direction, while in the radial direction we can identify three main regions characteristic of the two recirculation zones, internal, external and of the swirler flow. Furthermore, it has been noticed how the fuel jet which takes place at alternate vanes of the swirler does not determine persistent heterogeneity of the flow in the tangential direction.

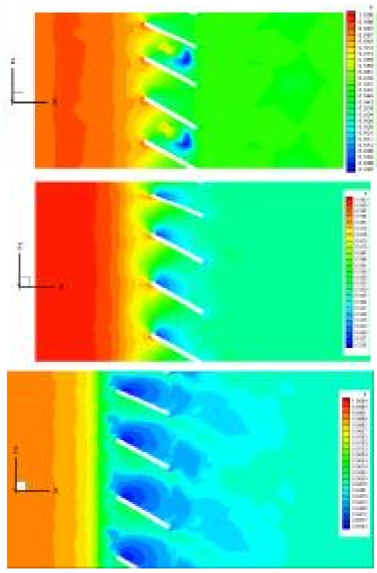


Speed vectors at middle plane of the swirler

Hub

middle radius

Tip



Static pressure

Fig. 5 Results of the fluid dynamic simulations on cylindrical sections inside the swirler

Figure 5 shows a synthetic description of the flow on blade-to-blade surface of the swirler. The blades of the swirler in these modification prototypes are realized with flat plates. It can be noted that the blades tend to stall on the side of depression immediately downstream from the leading edge. Moreover, the region at low pressure located on the suction side of the blade extends itself towards the tangential and axial direction, passing from the hub to the tip of the channel. The maps taken at the blade's hub clearly show the injection jet of the fuel with the beginning of a high pressure area approaching the jet followed by a depression region immediately downstream of the same. In this case the penetration of the jets seems clearly rather low. In conclusion, the fluid dynamic simulations allowed to define and verify the design modifications introduced to the injection system from an aerodynamic point of view. The best solution obtained has then been implemented and tested further using CFD and full scale tests for the whole combustor arrangement.

Results of the simulation of the complete combustor

The configuration singled out as the most promising on the basis of the previous phases was the object of further detailed investigations, including the complete combustion system. The following figures show the solid model and the mesh on a longitudinal section next to the liner holes axes. The solid model was rebuilt as accurately as possible, starting from the swirler outlet section, considering the primary, secondary and dilution zones liner holes and the openings for the film cooling of the liner. The objective of the simulation was to evaluate the interaction of the new injection system with the aerodynamic operation characteristics of the combustor, also evaluating the influence of a further blow-by for the film cooling of the dome, needed to smooth the peak of the metal temperatures found during the experimentation.

The simulations were carried out posing as inlet boundary condition to the combustor the swirler outlet section. Here the ideal angle of the flow has been imposed by the swirler blading, 30° angle. Higher angle values were not considered to avoid the stall of the blades of the swirler. Larger angles would require curved blades and consequently an undesired greater manufacturing complexity. The computations were carried out imposing the design ratio Dp/p at 0,05. The total pressure value for the flow intake section from the swirler was reduced in order to take into consideration of the pressure loss introduced by the swirler itself, which was estimated from the previous computation. The great care in the geometrical design and mesh generation may be appreciated in Figure 6.

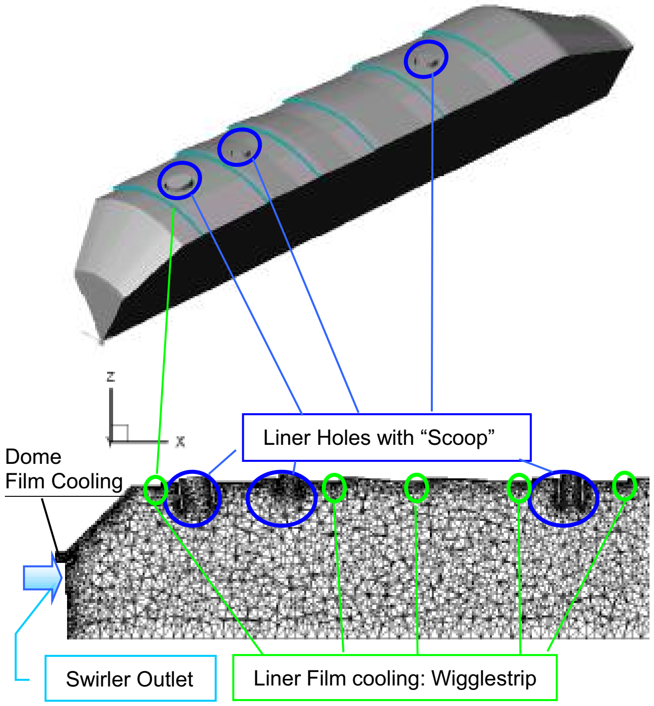
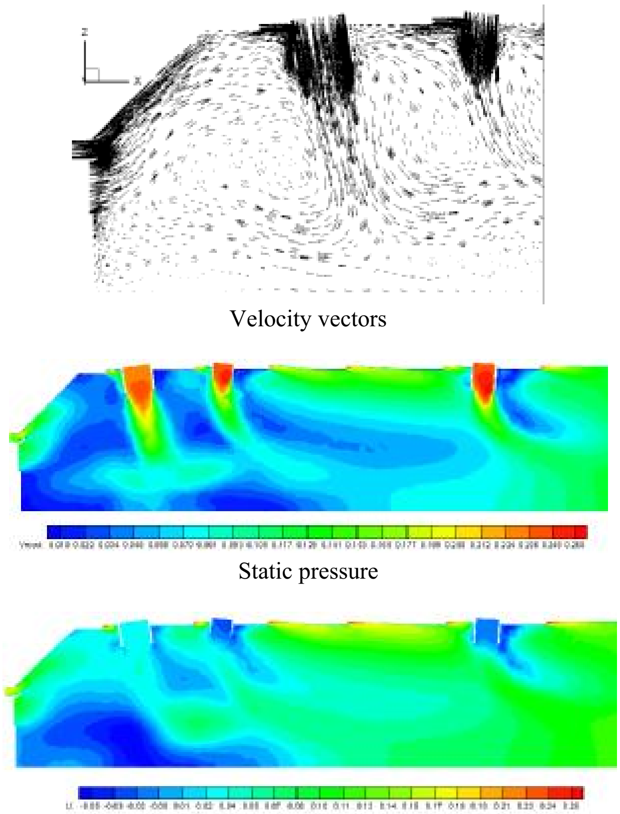


Fig. 6 Solid model and calculation grid for the combustor

Figures 7 show the results of the numeric simulation taken on a longitudinal section, passing across the axis of the liner holes. It can be noted that the aerodynamic field generated is typical of an old generation combustor, with a rather flat separation between the primary, secondary and dilution zones. In the primary area we have a typical toroidal recirculation zone, which involves the air coming from the swirler and part of the air coming from the primary holes. The extension of the recirculation zone both in the axial and in the radial directions is well in sight as can be seen from the maps of the speed vectors and of the axial component. The direction of the primary air jets and their penetration seems affected by the angle of the flow coming from the swirler. The primary jets take a slightly curved form towards downstream, while for the original model the jets took an approximately straight direction coincident with the axis of the hole.

The air blow-by realized with an axial input placed over the external diameter of the swirler for the cooling of the dome does not seem to determine operating problems since it stays flat against the wall. It then mixes itself with the edge of the blow-by main vortex, as can be noted from the maps of the speed components. On the other hand the presence of this “cold” flow could help the formation of unburnt exhausts. Weak flow separations zones take place in the region between the primary and the secondary holes and downstream of every liner holes for inlet air since equipped with scoop. The penetration of the secondary jets seems too high, the fresh air flow is concentrated in the region of the combustor’s axis, most likely leaving the adjacent areas at higher temperatures since the flow coming from the primary area does not go through an appropriate mixture with the secondary air.

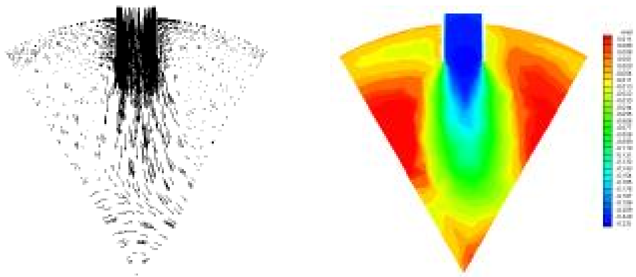


Velocity vectors

Static pressure

Axial velocity components

Fig. 7 Results of the aerodynamic simulation of the combustor on the longitudinal plane



Velocity vectors

Radial velocity component

Fig. 8 Results of the aerodynamic simulation of the combustor on a cross plane in correspondence with the axis of the primary holes.

Figures 8 show the results taken in a cross section in correspondence with the primary liner holes, from which the behavior of the primary air jets can be investigated. The analysis of the maps shown highlights that the aerodynamic field in these sections is characterized by three main regions: the air jet from the hole, the central blow-by area with tangential speed component and the peripheral region which, on the contrary, shows higher values of the axial speed component. Numerical simulation repeated for different swirler flow angle shows as the angle of the swirler can affects the penetration of the jet and the extension of the central recirculation zone in radial direction. The flow close to the wall of the liner takes the maximum speed values for the effect of the air flow coming from the film cooling device (wiggle-strip).

In short, the aerodynamic study of the modified combustor, on the whole, was used to assess a regular operation. The analysis of the results suggested a series of further detailed modifications for the optimization of the combustion system. Among all, it is worth reminding that the swirler angle and the internal diameter of the same, which affect the dimension and the intensity of the recirculation zone. Further investigations could address the “scoop” devices of the liner holes and the dimensions of the holes themselves. In this sense the role of guidance of the numerical simulations with respect to the further design modifications activity offers promising results.

FULL-SCALE TESTING

From the atmospheric test-rig experiments coupled by 3-D aerodynamic simulations, two modified configurations were selected. They were tested on the ENEL test bench at Sesta (Benelli et al., 2003), Figure 9.

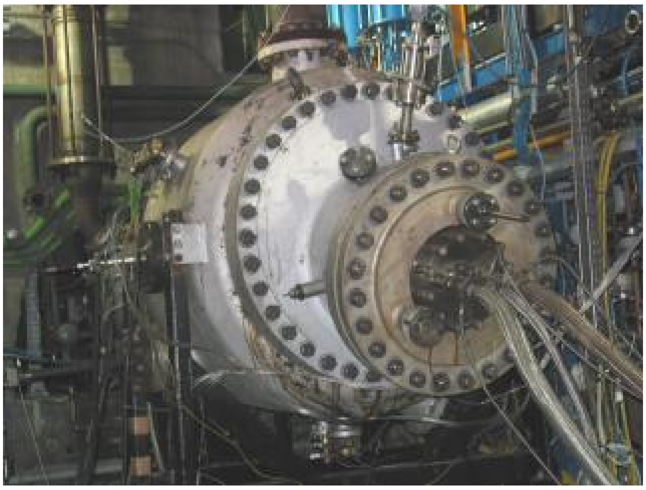


Fig. 9 The combustor at the Full pressure test rig of Sesta

This plant allows the combustor to be tested at full scale, even with water injection, thus reproducing the actual operating conditions. In particular three configurations were tested: Original Configuration, Modified Configuration A and Modified Configuration B. The modified configurations were equipped with the new injection system characterized by the axial swirler and by the fuel injection in the vanes among its blades. Configuration A differs from Configuration B due to an array of fuel injection holes placed upstream of the axial swirler and due to a different inclination of the main holes axes. The results of the measurements obtained from the tests are: the emissions’ curves, the metal temperatures and the time-history pressure fluctuations in a wide range of the thermal load. The pollutant emissions measurement is made with chemical analysis benches produced by Siemens. Every gas analysis system is made up of a probe for the sample, a filter, the suction pump and the O2, UHC, CO, CO2 and NO analyzers with converter for NO2s.

The sampling probe, placed at transition piece outlet section, is of the multipoint type (5 points taken in the tangential direction). For the measurement of the temperatures of the metal wall of the combustor and of the exhaust, were performed by thermocouples “K” type (Nichel – 6% Aluminum) which are characterized by a measurement range between 400 and 1.200°C with an accuracy of +/- 0.5%. The fluctuations of pressure were measured by means of a piezoelectric pressure transducer, connected to an amplifier, spectrum analyzer and a personal computer for recording and post-processing (Fast Fourier Transform) of the data. Each test was preceded by the chemical analysis of the fuel gas used, by means of a gas-chromatograph.

The high pressure test rig, utilized for combustion testing, is presented in Figure 10, where the combustor completely installed and an internal viewing system for image diagnostics are also shown.

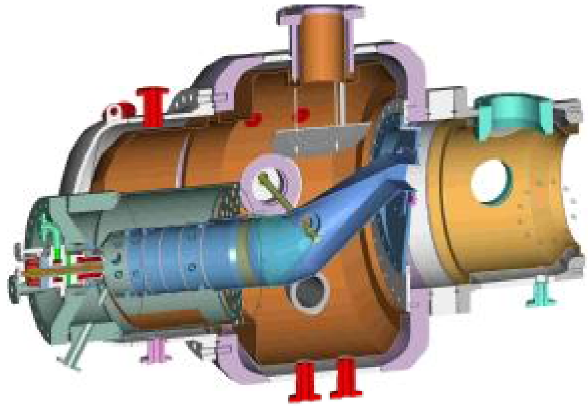


Fig 10 High pressure test rig

Results and discussion

The comparison between the different models of the combustor tested is mainly carried out on the basis of the emissions’ values, (CO, NO_x) @ 15% O2 at ISO conditions, of the consumption of water, of the temperatures of the metal and of pressure fluctuations. For the operation in “Dry” state, the curves of the emissions of nitrogen and carbon mono-oxide can be compared, while for the tests with injection of water the values of the water flow needed to reduce the levels of emissions of NO_x below the 50 ppm limit are compared considering, at the same time, the emissions of CO. With Dry operation the modified models A and B bring a good reduction of NO_x emissions. Figure 11 shows the non-dimensional NO_x emissions curves in the range of 25-110% of the equivalent electric load. Model A determines a reduction of about 25% of NO_x at full-load. The reduction is present in the whole operating field; it can be noted how this reduction decreases as the load increases passing from a reduction of about 44% at the 25% of the thermal load to a reduction of 25% circa at the highest (75-110%) load values. For the modified model B the emissions’ reduction is slightly lower in the whole operating field being tested (-20% at full load). This behavior can be credited to the better air/fuel mixing due to the double injection holes row (mod. A) and by the different tilt angle of the main fuel injection holes which supply the most part of the fuel flow rate (75%), thus confirming the results previously obtained on the test bench at atmospheric pressure of the Livorno testing area. The performance improvement in terms of NO_x emission is shown on the whole working range; at 100% of the thermal load the reduction with respect to Model B, is around 8 ppm, equivalent to 6.3%.

The emissions of carbon mono-oxide are much lower for all models in Dry operation. For values of the thermal load over 50%, the CO in the exhaust is lower than 1 ppm. At 25% of the load the two modified models show values of about 63 ppm compared to 10 of the baseline model. In the wet operation the CO emissions rise considerably. At 100% of the load we can read 3.7 ppm for the original model, 11.4 ppm for model B up to 43.1 ppm for model A. Considering that the CO emissions are very low in Dry operation, these increases are due to the influence of the water injection and to the water injection methods and consequently to the air-fuel-water mixing. In particular for the modified models there is also highlighted a considerable increase in the emissions moving towards low values of the thermal load, unlike the original model that keeps the CO below 8 ppm up to 75% of the thermal load, even in “Wet” operation.

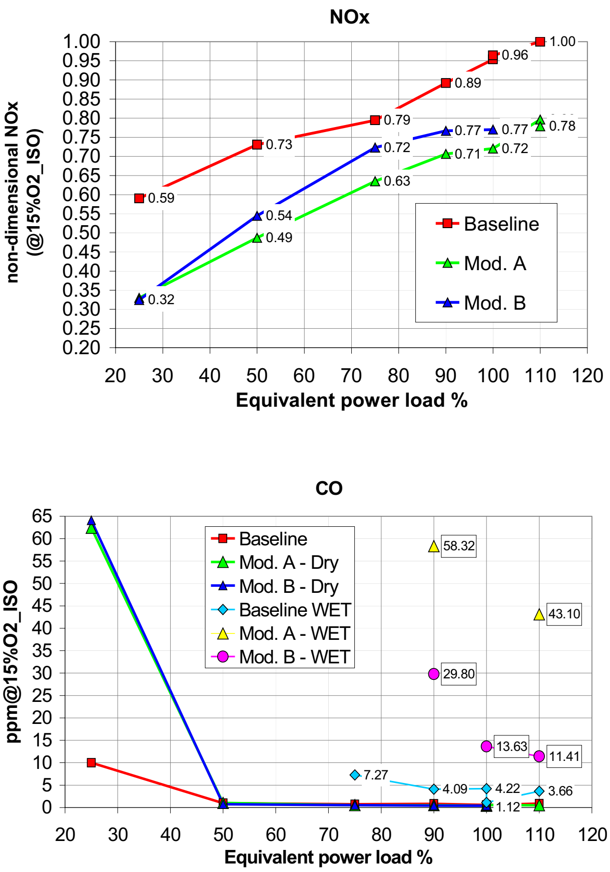


Fig. 11 NO_x and CO (@15%O2_ISO) emissions curves tested at full pressure conditions

Some suggestion of particular importance comes from the comparison between model A and B in Wet operating conditions. Model A, which in Dry conditions gave good results, is not capable of a satisfactory performance in the case of water injection. The water injection is performed together with the injection of fuel and in absence of a previous atomization process. Given this configuration of water injection, it must be noted that model A presents high values of CO emissions (58.3 ppm at 90% and 43 ppm at 110% of the equivalent power load) and requires high water flows 0,34 and 0,39 kg/s respectively for 90 and 110% instead of 0,32 and 0,40 kg/s for the original model, in order to contain the NO_x down to 50 ppm, Figura 12. Probably the diameter and the axes inclination of the injection holes are not suitable for the water injection directly from the gas injection holes. All this implies that the water injected is sent to colder regions of the combustor, thus determining a high formation of CO without contributing to an effective cut in nitrogen oxides. Viceversa the Modified model B with injection holes axes inclination of 30° and with a larger diameter determines a considerable improvement in the wet operation, even starting from slightly worse conditions of emissions in Dry operation. In this case the consumption of water at 100% of the thermal load is reduced by about 19,4% (-0.07 kg/s). The emissions of CO rise, even if they stay at acceptable levels, 30, 13.6 and 11.4 ppm respectively at 90, 100 and 110% of the load. In conclusion the different dimensions and inclination of the holes of model B positively affect the effectiveness of water injection. Figure 12 shows also the water - NO_x diagram at full load. With regard to the UHC emissions, these follow the pattern of the CO emissions, but assume much lower values in all configurations.

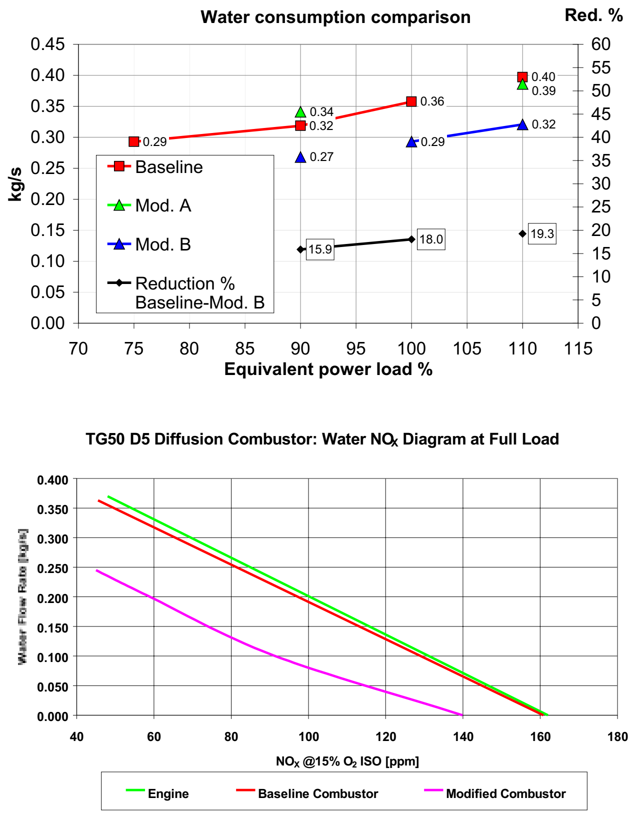


Fig. 12 Water consumption comparison and Water - NO_x diagram tested at full pressure conditions for full load operation

Figure 13 shows the curves of the metal temperatures for the three models in operating wet conditions at 110% of the load. The temperature values shown are the maximum among the ones taken in a same section. The curves show values that are all very similar.

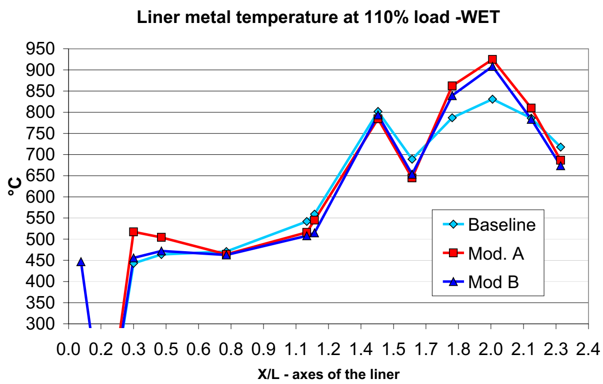


Fig. 13 Maximum temperature curves of combustor wall, x/L non-dimensional co-ordinate L = liner lenght

The result from the spectrum analysis of the pressure fluctuations indicates that in wet operation the peaks of pressure are higher than the ones obtained in Dry operation. The maximum values were recorded for the original model. For the modified models; a considerable decrease can be noted in the peak values of the pressure fluctuation as well as an increase of the frequencies at which they occur. Modified model B is the one, which presents lower peak values, from 280 to 150 Pa going from 90 to 110% of the load at frequencies between 50 and 85 Hz. With regard to the temperature profile at exhaust, an accurate value of the pattern factor was not detectable since the measuring section was equipped with only three thermocouples. In any case the maximum unbalancing measured among the three temperature values taken is not higher than 10%.

Besides verifying the results obtained, The set of full scale tests provides important information related to the next steps in the development of the Fiat Type 50 D5 STD combustor. The tests confirm the considerable influence of the geometry of the holes of the injection system, which determines the characteristics of the jet and of the cross-flow with the air flow rate, and finally on the combustor performances.

CONCLUSIONS

The study for the improvement of the combustion chamber for gas turbines, carried out in collaboration with ENEL- Ricerca Team was tackled with an integrated design methodology which includes the application of numerical/experimental tools of different type and complexity. Mono-dimensional models were applied for the preliminary design, while tri-dimensional "full Navier-Stokes" (CFD) models were applied for detailed 3-D analysis of the aerodynamics of the combustor. The CFD code used proved to be suited to the resolution of the flow for this type of geometries. In particular the mesh generation code of the hybrid non-structured type, seems to be particularly suited for the discretion of complex domains like the ones defined by the geometries of the gas turbines' combustors. The combustion process was investigated mainly with experimental type tools, with tests on the complete combustor, both at atmospheric pressure and full scale test-rigs. The activity, on atmospheric pressure and full pressure test bench, showed furthermore that the combustion experimentation at atmospheric pressure is capable of giving useful and coherent information for the development of combustors for gas turbines (Martelli et al., 2001).

The numerical and experimental tools considered were applied to the study of modifications to the combustor that equips Fiat TG50 D5 STD machines, aimed at limiting pollutant emissions. The study led us to define a series of modifications for the new injection system with partial pre-mixing, to be used in place of the original injection system with diffused flame. The combustor equipped with the modified injection system guaranteed, during full pressure tests, a reduction in emissions of about 25% in Dry operation and of about 19% of the water mass flow rate requested for the NO_x pull down in wet operation, keeping unchanged the original functionality of the combustor, object of the modification. The new injection system was developed by keeping the realizations costs low and by trying to keep to a minimum the intervention on the liner. The results of the activity carried out allowed us to define further modification actions for the improvement of the operation both in dry and in wet conditions. From a methodological point of view, it can be asserted that the work procedure based on the integrated use of experimental activities and numerical simulations, proved to be appropriate for facing and solving problems related to the retrofit of gas turbine combustors.

AKNOWLEDGEMENT

The authors wish to thanks the ENEL-Ricerca Team of Livorno: L. Carrai, S. Calvetti, F. Ferrari, R. Micheletti and F. Salimbeni as well.

BIBLIOGRAPHIC REFERENCES

Adami P., Martelli F. and Michelassi V., 2000, "Three-Dimensional Investigations for Axial Turbines by an Implicit Unstructured Multi-block Flow Solver", *Proceeding of IGTI Conference – Munich* , ASME Paper 2000-GT-0070.

Adami, P., Martelli, F., "Numerical Computation of Turbulent Non-Premixed Reacting Flows in Combustion Chambers" GTSJ International Gas Turbine Congress 1999 Kobe, Japan.

Benelli G., De Michele G., Trebbi G., 1996, "Combustori per Turbogas" ATI - *Quaderni di Termotecnica n.1*-1996.

Benelli G., Barbucci P., Mariotti G., Tirone G., 2003, "Retrofittable solutions for combustion systems to improve GT emissions and performances", Powergen Europe Dusseldorf (D) 6-8 May.

Feitelberg A.S., Tangilara V.E., Elliott R.A., Pavri R.E., Schiefer R.B, 2000, "Reduced NOx Diffusion Flame Combustors for Industrial Gas Turbine", *Proceeding of 45th ASME - IGTI Conference – Munich, Germany, 8-11 May*, ASME-0085-GT-2000.

Holdeman D., 1972, "Correlation for the Temperature Profiles in the Plane of Symmetry Downstream of a Jet Injected Normal to a Crossflow", *NASA Technical Note D-6966.*

Lefebvre A.H., 1999, "Gas Turbine Combustion", *Second Edition Taylor&Francis: Philadelphia 1999.*

Jacob T., Duane A., Razdan K., 1995, "Development and engine testing of a Dry Low Emissions Combustor for Allison 501-K industrial Gas Turbine Engines", *Proceeding of IGTI Conference – Huston, Texas*, ASME paper 95-GT-335.

Martelli F., Riccio G., G. Benelli, D. Cecchini, L.Carrai, 2001, "Scaling from Atmospheric Pressure Rig to Full-Scale Pressure for the Emission Measurements from a Gas Turbine Combustor", *Proceeding of IGTI Conference – New Orleans*, ASME Paper 2001-GT-0070.

McVey J. B., Padget F.C., Rosfjord T.J., Hu A. S., Peracchio A. A., Schlein B. and Tegel D. R., 1992, "Evaluation of Low NOx Combustor Concepts for Aeroderivative Gas Turbine Engines", *Proceeding of IGTI Conference – Cologne, Germany*, ASME paper - 1992-GT-0133.

Riccio G., Adami P., Martelli F., Cecchini D., Carrai L., 2002. "Improvement of Gas Turbine Injection Systems by Combined Experimental/Numerical Approach", ASME Paper 2002-GT-30101, *Proceedings of 47th ASME IGTI Congress, Amsterdam, The Netherlands*

www.Centaursoft.com