THERMAL-FLUID-MECHANICAL MODELING APPROACH AIMED TO DESIGN A TRAPPED VORTEX COMBUSTION CHAMBER OPERATING IN MILD CONDITION

R. Di Canosa*, A Di Nardo*, C, Mongiello, G. Pesce****
carmine.mongiello@enea.it
*ENEA National Agency for New Technologies, Energy and Sustainable Economic Development, Casaccia Research Centre.
**ENEA National Agency for New Technologies, Energy and Sustainable Economic Development, Portici Research Centre.

Abstract

The problems related to energy efficiency and polluting emissions reduction direct studies toward low-impact solutions, to obtain stable and adequate operating condition of the plant. MILD combustion is one of the methods used to achieve these aims, allowing a drastic reduction of polluting emissions, in the first instance of the NO_x. A careful design of the combustion chamber ensure a correct exhaust gas recirculation, which is essential for the flameless process stability. To ensure the desired evolution of the fluid flows reagents is indispensable to identify the geometric configuration and the appropriate allocation of suitable courses, as well as the optimal arrangement of the nozzles for the air and fuel admission. In addition, the deformability of the walls of the chamber have to be considered: the thermal and fluid dynamics stress produce some structural deformations and in some case they may change trends of flows and perturb the stability of the combustion.

Purpose of the study

The study focuses on a Vortex-Trapped type combustion chamber, which, as the name suggests, uses the high kinetic energy from the incoming fluid to generate a vortex; in this way the recirculated exhaust gas reach the desired dilution of the combustion mixture. The device has an annular axisymmetric geometry, because the burner is intended for use in gas turbine installations. The fuel studied during the research was methane, but the ultimate goal is to set up a template on which to operate by changing the fuel: choosing a mixture with a lower heating power, fluid flows must be increased in case of power plant maintenance and the stress on the walls becomes greater. So the need to identify the incidence of the thermal, fluid dynamical and pressure stress on the resistance properties of the burner, in the new operating condition.

Objectives

The primary target to gain is obtaining the MILD combustion. The realization of such an operating condition is proved by a rather uniform temperature field and by the lacking of an high temperature region associated with a flame front; almost simultaneously, by low concentrations of nitrogen oxides and unburned exhaust [1]. As previously mentioned, the problems related to thermal and mechanical stress on the structure may impair the devise working, so we must abandon the assumption of infinite rigidity of the walls, we have to analyze the stress and the strain of the structure and, ultimately, we shall ensure an adequate cooling of the burner. The cooling is done using a flow rate of exhaust gases recirculated in proximity of the inside wall of the combustion chamber and a flow from external ambient is blown into an outside interspace, whose surface has been suitably finned in order to maintain the temperature of the wall at about 1100 K, improving the heat exchange. Finally, it is appropriate to analyze the performance of all variables considered in the case of using fuels such as ethanol, or syngas, thus adjusting the flow rates necessary to maintain the same thermal power plant. To achieve the attainment of MILD combustion stability [5] [6] [7], it is essential to increase the residence time of reactant flow within the chamber and to obtain the correct mixture dilution energizing the exhaust gas recirculation. At the same time is fundamental turn away from the wall the flame front and reduce the concentration of fuel centrifuged into this area, so that combustion efficiency is improved. The first change was done to the geometric configuration of the room: the walls were bended, thus the evolution of the vortex is favored. A part of the already burning gas is driven to the nozzles of fresh charge admission and some benefits in mixing, dilution and mixture heating are obtained. The use of a short divergent as exhaust discharge duct allows a pressure recovery that increase the residence time of the gas into the combustion room, however, the rapid reversal of direction of travel flow causes a low pressure area in the vicinity of the spout inside the room and there is a recirculation zone and fluid losses are added. Obviously these losses are harmful to the plant functioning.

Method

The study starts from a previously tested configuration in which they obtained a discrete gas recirculation, which was characterized by excessively rapid fluid evolution: the rapid discharge of the ignited mixture prevented a sufficient spread of chemical species and the completion of the reactions which are sources of nitrogen oxides [1]. Furthermore, as the high kinetic energy of the current induced densification of the fuel and the ignition of reactions in a region very close to the chamber wall, two types of complication resulted: the development of the chemical wall reactions, which hinder the completion of the oxidation reactions, and the thermal stress worsening. The reached temperatures make more relevant and difficult the wall cooling. In order to obtain better performances, a new more efficient configuration was found after some simulations done for different layouts.

Considering the new configuration presented some benefits, the identified problem of the excessive heating of the chamber walls remains and structural deformations were large. To create a meatus of colder gas near the wall was the solution studied: thanks to a bypass, flue gas were taken from the exhaust nozzle and inflated into the combustion chamber. These nozzles are located near those for fresh air and fuel, but also closer to the burner structure. Thus, it is possible to give a high vorticity to the fluid in the chamber driving the fluid mass in rotation by the current inert and without adopting elevated velocities for the burning mixture; the fuel mixture thickens in a straightly central region far from the wall and it stays there confined for a greater amount of time. The result is that the temperature does not grow excessively during combustion and the chemical process completes and pollutions are reduced, using a recirculated exhaust flow of comparable magnitude with the primary air [2] [3] [4]. It is also a designed the presence of an outside cavity where a flow of ambient air circulates and warms, holding the temperature of the walls. A finned surface is chosen to maximize the cooling of the structure, without generating excessive heat loss in the combustion process and containing pumping costs because of the lower mass flow required. The size of the blades is defined by a parametric investigation optimizing the heat exchange and keeping the best compromise between the required flow rate necessary for the cooling and the arising of the pressure losses following the reduction of the pass between wings. To improve fluid evolution in the room some nozzles dedicated to a secondary fuel and fresh air flow rates were added, this choice derived from the analysis of the of chemical species concentration fields, because sometimes an excessively lean mixture were created and the flame lift-off were noted.

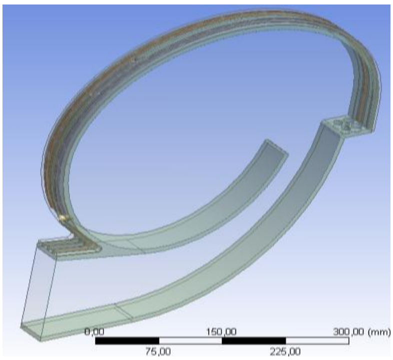

Figura 1. Geometry

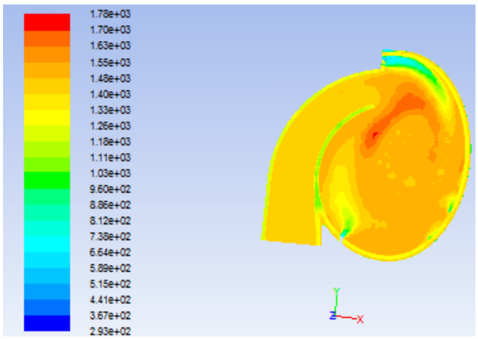

Figura 2. Temperature field

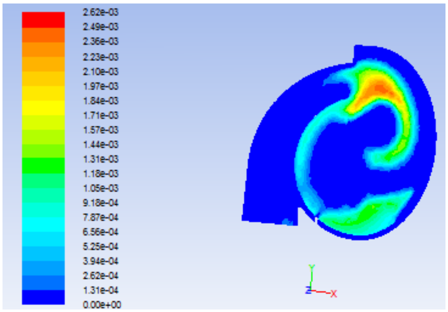

Figura 3. OH Mass fraction field

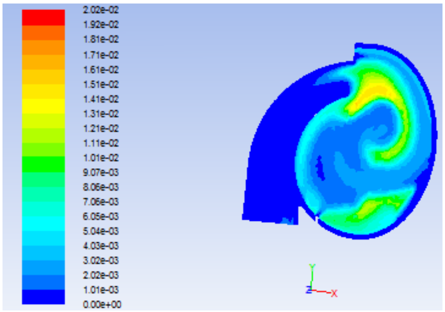

Figura 4. CO Mass fraction field

The stress analysis is conduct on a cantilever structure that is constrained in the final section of the exhaust pipe, the purpose of this choice is in the minimization of the stresses allowing thermal expansion. The stress related to the weight of the body has been neglected in order to focus the analysis only on the working conditions effects. The kinds of stress mainly affecting the body are related to the inside pressure field, that it is kept of some bars to prevent infiltration of unwanted outside air: the flameless combustion is extremely sensitive to environment conditions and to change operating condition and dosage may endanger the process. Some of the actions are related to the velocity field: the current covers the body with high-speed flows and it is obviously diverted by the wall, thus the fluid dynamic stress is generates on the fluid-structure interface; finally the most relevant stress is originate from the thermal gradient between the inside of the chamber and the cavity through which flows the coolant. It is immediately apparent that the presence of this magnitude permits to reduce the temperature difference by reducing the stress in the wall. The study was conducted on different types of material: stainless steel, aluminum alloy and titanium alloy. We note that from the fluid dynamics standpoint we didn’t observe any significant change in all three cases. One might expect some minor differences in the combustion process induced by thermal losses in the room: the different values of thermal conductivity may modify the chemical development of the reactions, but thanks to the rapid dynamics of the oxidation phenomenon and to the proper characteristics of temperature uniformity of the MILD combustion [2] [5] [6] [7], there is no significant effect. Certainly in all three cases are detected deformations of different entities, but the cantilevered layout of the system gives rise to roughly isotropic movements; slightly different is the arrow inside the beak, but even if the radius of curvature of that wall slightly increase and the gases are a little less well driven to the power nozzles, there is a corresponding reduction in the flow section of the discharge that slows down the process of removal of the exhausts, so the same operating conditions are almost ensured. The most severe conditions of thermal stress were feared for the inside beak, but examining the results obtained and looking at the temperature range, one notice that in that element the temperature is maintained low enough by the secondary fresh air supply and in the rest of the structure occurs a desired temperature near to 1050k. The analysis takes advantage of the hypothesis of elastic superposition of states, because the deformations are included in the elastic range of strain, so it is possible to examine separately the contributions of stress and deformation. It is revealed that the stress becomes more significant values corresponding to the curved walls of the chamber, which have to drive and change the trajectory of the flow. As on the other hand regards the, the main thermal effects are obviously related to the regions in which occurs those points which are mainly exposed to the flow, such as the beak, and sections which are subordinated to high gradients. Comparing the results one can notice that due to the superposition of the effects the displacement with the higher magnitude occurs obviously in the farthest point from the joint section, where, furthermore is revealed a significant thermal stress. The dominating stress is obviously the heating one, to which the 98% of the total stress is attributable, and corresponding similar percentage of deformation accordance to the hypothesis of elastic linearity of the material. It is still observable that in these operating conditions it remains sufficiently far from the value of the yield strength, the assumption of a trend in the elastic stress-strain ties is therefore justified. It is detected a main bending nature of the solicitation, relative to the axis of symmetry of the whole component in its development road. In fact, considering a cross-section plan, it can notice a symmetrical pattern of stress in the section, as expected from the model of stress Juransky. Finally it should be revealed that due to the large number of cells used during the process and the reduced power of a single machine, the effects along the edges tend to affect the results, which can, however, be significant in the central area of the body, certainly the analysis of the entire component developed in the entire 360° would provide more accurate solutions.

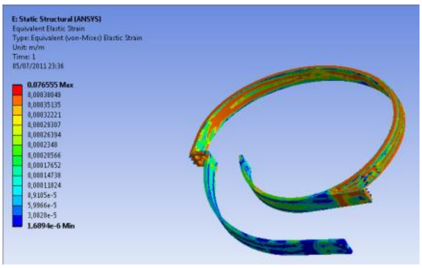

Figura 5. Elastic strain

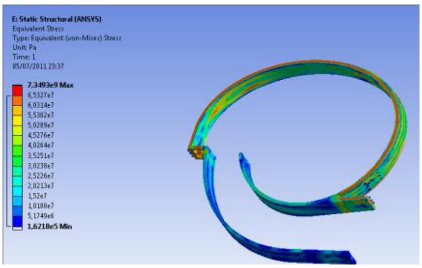

Figura 6. Equivalent stress

Tools

For the solution of the problem we used the FEM ANSYS 12.1. software implementing in it some kinetic schemes, which are able to model the reactive phase, running appropriate algorithms. The model consists of a single portion of the combustor in which were used 1.5 E+6 cells, an amount related to the need of a contemporary mesh for the fluid and the structural region, there were essential to maintain a coincidence of size between finite element at the interface of the wall, to allow the calculation of heat transfer from the algorithm of the software.

Furthermore, to make the analysis sufficiently close to reality, it was identified the need to dispose three rows of cells in the interior of the thickness of the combustion chamber. It was also identified the require to use the computing power of more processors working parallel: during the previous analysis inconsistencies in the patterns of the tensions within the structure were found, probably due to an insufficient refinement of the mesh in the region of the edges; increasing the number of cells and densifying the grid inside the thickness, will allow to reduce the problems on board. In order to make the fields of temperature, pressure and speed, as well as the distribution of species during the combustion process, closer to reality it can be identified the need to adopt more complex kinetic schemes, choosing algorithms to implement in the calculation software, that involve a greater quantity of reactions and species, assuring a better performance. Furthermore, it is certainly an advantageous to reduce the computation time for the same model computation and improve the accuracy densifying the mesh. By changing the type of fuel, in the specific case of ethanol, there is a much more complex chain of reactions that leads to the complete combustion of the mixture, including a larger amount of intermediate products, it is considered that a Chemchin algorithm useful for the modeling of a reagent flow obtained by mixing air and ethanol comprises on average 250 reactions, requiring much higher computing capabilities.

References

[1] W. Bartok, V.S. Engleman, R. Goldstein, E.G. del Valle, “Basic Kinetic Studies and Modeling of Nitrogen Oxide Formation in Combustion Processes”, *AIChE Symp Ser 68(126):30,1972.*

[2] M. Katsuki, T. Hasegawa, “The Science and Technology of Combustion in Highly Preheated Air”, *2-Symposium (International) on Combustion, The Combustion Institute, Pittsburgh, 1998, pp.3135-3146.*

[3] N.K. Rizk, H.C. Mongia, “3-*D Emission Modeling for Diffusion Flame”*, Rich/Lean, and Gas Turbin Recirculation, PhD Thesis, Technion, Israel, 2002.

[4] J.H. Seinfeld, *Atmospheric Chemistry and Physics of Air Pollution*, J. Wiley & Sons, New York, 1986.

[5] M. Derudi, A. Villani, R. Rota, The Influence of Operating Parameters on Mild Combustion Sustainability, *Dipartimento di Chimica, Materiali e Ingegneria Chimica “G. Natta” / CIIRCO – Politecnico di Milano, Milano – ITALY.*

[6] Mara de Joannon, Antonio Cavaliere MILD Diffusion-Ignition Lean Premixed Combustion, *Istituto di Ricerche sulla Combustione Università Federico II, Na*poli

[7] A.Milani, Mild combustiontechniques applied to regenerativefiring in industrial furnaces, *Centro sviluppo materiali – experimental station of combustion, Genova.*