Numerical Simulation of an Industrial Radiant Burner

G. Scribano, G. Solero, A. Coghe

Dipartimento di Energetica, CIIRCO – Politecnico di Milano, Milano, Italy

ABSTRACT

The present paper reports the 3-D CFD modelling of an industrial swirl radiant burner (12.8 – 18 kW) performed by means of a commercial code (FLUENT). The first step was the validation of the code through a detailed comparison with experimental velocity data acquired under isothermal flow regimes for different operating conditions. On this basis the reactive characterization was carried out and the thermal and fluid dynamic flow patterns were compared with the experimental measurements. The k – ε RSM (Reynolds Stress Model) model was used for Navier – Stokes equation closure and the air – methane reaction was modelled with the 2 step reaction approach with Finite Rate – Eddy Dissipation model. For the radiative characterization it was chosen the DO (Discrete Ordinates) model. Moreover, the simulation of a modified burner geometry with internal EGR was performed and confirmed the experimentally observed reduction of pollutant emissions.

INTRODUCTION

Industrial radiant burners have been studied in the past mainly from the experimental point of view and typical results are available in [1]. The main features of this type of burner are the presence of a swirl component in the reacting air flow to improve mixing and flame stability and the confinement of the flame inside a tube. Detailed experimental analysis of the mixture formation, combustion evolution and degree of internal EGR (Exhaust Gas Recirculation) generated in these burners is quite difficult to perform and information on numerical simulations were not found in the open literature. The present paper reports on a systematic attempt to use the CFD tool, validated by experimental data, to analyse the combustion process and improve the design of the burner. The simulation has been performed through a commercial CFD code (FLUENT) and the validation was carried out by comparison with experimental data obtained in a model burner, where the velocity and temperature profiles were measured. After validation, the code was used for a parametric analysis of the effect of the degree of internal EGR on the reduction of pollutant emissions.

CFD SIMULATION OF THE BURNER

The burner geometry is depicted in Fig.1. The burner is fuelled by natural gas and air in a coaxial configuration characterised by three gas outflows at 120° and the swirled air injected through a annular opening with six lobes. On the central hole a pilot premixed flame is stabilized. Combustion is confined in a long flame tube and the hot products are transported to the exit in the gap between the inner tube and the external radiating one. A small fraction of the exhaust gases should recirculate in the main combustion region through a variable small gap between the burner efflux and the flame tube inlet. This recirculation mechanism was found very poor in the original burner configuration and a more effective solution was selected after a detailed experimental and numerical analysis. The final solution is based on the addition of a pre-combustion chamber terminated by a converging nozzle acting as a mixing/ejector to improve exhaust gas entrainment into the flame tube (see Fig.2). The periodicity of the efflux section of the burner allowed to model only a wedge of 120° adding periodic boundary conditions on the two wedge sides. The isothermal condition was analysed by using a grid of about 8x10^5 hexahedral cells for the fluid volume, while for the...
reactive conditions the number of cells was increased to $9\times 10^5$. The Second Order Upwind Scheme was adopted for the discretization and the SIMPLE algorithm was used for the pressure – velocity coupling to solve continuity and momentum equation. Steady state simulations have been performed with the turbulence modelled by the RSM (Reynolds Stress Model) [2,3,4], since the gas turbulence in a swirling flow exhibits a non-isotropic nature.

For the combustion process a methane–air 2 step reaction scheme was used and the reaction rates were computed from the Arrhenius rate expression and from the Eddy Dissipation concept of Magnussen and Hjertager. Thermal radiation was included in the simulation using the DO (Discrete Ordinates) model. Details about the combustion and radiative models are in [5]. The case with internal EGR (Fig.2) was studied with the same code and a similar number of cells as the original case without EGR. After many tests, the pre-chamber and nozzle geometrical features were evaluated to obtain $\approx 40\%$ recirculation, following the design criteria discussed in [6].

**RESULTS**

Fig.3 presents the comparison between the radial profiles of the axial and tangential velocity components (experiments and simulations) for the isothermal flow at axial distance $x/R = 0.26$ from the burner efflux. The results for the reactive case at the same axial distance and
equivalence ratio $\Phi = 0.85$ are reported in Fig.4. It can be noticed that the agreement between the measured and computed velocity profiles is reasonably good in both the isothermal and reactive cases, although the differences are more pronounced in presence of combustion. Comparison of the axial components indicates that the simulation fails in the centreline region giving rise to lower velocities and consequently the amount of the recirculating flow is overestimated [2]. Regarding the tangential profiles, the discrepancy is mainly on the magnitude of the velocity in the external zone; the difference may be due to the spatially averaging effects of vortex core precession in the measured values [4].

![Fig.3 - Comparison of the numerical and experimental axial and tangential velocities. Isothermal flow; axial distance from the burner efflux, $x/R = 0.26$](image1)

![Fig.4 - Comparison of the numerical and experimental axial and tangential velocities. Reacting flow, $\Phi = 0.85$; axial distance from the burner efflux, $x/R = 0.26$](image2)

The main validation test of the code predictions under the reactive regime was the comparison of the average temperature distribution evaluated along the burner axis with the values measured by means of a long thermocouple inserted from the end of the cover tube of the burner. Fig.5 compares the measured temperature axial profiles, for the condition of 12.8 kW, $\phi = 0.85$ and the standard burner geometry, with the computed values for both the reference case and the modified one with enhanced internal EGR (see Fig.2). Near the burner efflux ($x/R < 0.25$; $R =$ radius of the burner exit) the experimental values are higher than the simulated ones, the difference probably due to an incorrect simulation of the mixture ignition mechanism. In the main combustion region ($x/R \approx 0.25 \div 10$) the simulations overestimate the measured temperature, likely because of the simplified reaction model adopted. In fact, at high temperatures, the pyrolysis of CH₄ plays an important role and this endothermic reaction which decreases the temperature cannot be simulated by the reaction model used [7]. This consideration is partially confirmed by the CO concentration levels, with the measured values higher than the simulated ones. In this zone the presence of EGR is effective in reducing the
peak temperature, which will result in lower NOx emissions. In the post flame zone \((x/R \geq 10)\) the agreement between measured and simulated temperatures is reasonably good.

\[
\text{Fig. 5 - Comparison of the numerical predictions of the axial temperature and the measured values along the burner axis. } P_t = 12.8 \text{ kW; } \phi = 0.85
\]

\[
\text{Fig. 6 – Computed radial temperature profiles for different axial distances from the burner efflux. No EGR; } P_t = 12.8 \text{ kW; } \phi = 0.85
\]

Fig. 6 shows the computed temperature radial profiles, for different axial distances from the burner efflux. Data refer to the original burner geometry and show that the average temperature grows with the axial distance, while the profiles indicate a trend similar to the experimental behaviour reported in [8].

The comparison between the radial temperature profiles with and without internal EGR are reported in Fig. 7, for two axial locations at the entrance of the flame tube \((x/R = 4.5)\) and far downstream \((x/R = 9)\). As already observed in Fig. 5, the temperature is lower with the internal EGR, with the highest values located in the inner zone where the reactive flow is confined by the entrained exhaust gases at lower temperature and with a lower oxygen
content. Alternatively, in the case without EGR the maximum temperature is found in the external region of the tube, due to the centrifugal effect of the swirled air flow. The numerical simulation allowed to estimate the EGR rates and for the reported case its value was about 37% of the reacting flow rate at burner inlet, very close to the design value of 40%. This value was found to represent a good compromise between the decreasing NOx concentration and deterioration of the CO levels. At higher EGR rates, low combustion temperatures and high diluent concentrations produce a substantial increase of the CO levels at the exhaust and reduced combustion efficiency. These results were predicted by the code and verified experimentally, at least qualitatively.

![Radial temperature profiles computed for the standard burner geometry and the modified one with internal EGR, at two axial locations.](image)

The NOx emissions were measured with conventional gas analysers. Sample values are reported in Fig. 8 for two nominal equivalence ratios and the same thermal power (Pt = 12.8 kW; φ = 0.75 and 0.85) for both cases with and without EGR. The measured values are compared with the numerical predictions, which result higher due to the higher computed temperatures, as shown in Fig.5. The relative decrease of the NOx level produced by the internal EGR (of the order of 50%) is accurately captured by the code and confirms the positive effect demonstrated by the experimental results. The quantitative difference between predictions and measurements is due to the higher computed temperature level, compared to the measured one, considering that the NOx model is strongly temperature dependent. The qualitatively agreement between the simulation and experimental data lies only in the correct prediction of the percentage of NOx reduction using internal EGR; in fact the reduction in real and numerical cases is about 45%.

**CONCLUSIONS**

The numerical simulation of a natural gas burner used for industrial applications has been applied to the improvement of its design in order to reduce pollutant emissions in the exhaust gas. The numerical code has been validated by comparison with experimental measurements taken under isothermal regime, specifically the velocity flow pattern. The CFD code has been later used to analyse the behaviour of a modified burner geometry optimised to obtain a consistent internal EGR. The solution proved to be very effective in reducing pollutant emissions, mainly NOx.
Although the isothermal flow is predicted with a reasonable degree of accuracy, the numerical simulations of the combustion regime present only a relative accuracy. In fact, the code is not able to predict the real values for reacting flow although the qualitative behaviour is satisfactory. The differences are mainly due to the simplified reaction mechanism and the radiative models available on the commercial code.

**Fig. 8: NOx emissions by numerical simulation for original configuration and with internal EGR**

**ACKNOWLEDGMENT**

The authors are grateful to the CITT Impianti s.r.l. for the financial support of this project which has been carried out under the responsibility of Ing. P. Gennaro (CITT Impianti s.r.l.).

**REFERENCES**


