Numerical strategies for fluid-dynamic and heat transfer simulation for regenerative chambers in glass production plants

C. Cravero, D. Marsano, A. Spoladore

Abstract— The thermal regeneration systems are vital components for the energy performance of a glass furnace. The heat transfer in a conventional regenerative process is a complex phenomenon: mixed convection during the cold period (air flux) and combined presence of radiation ad forced convection during the hot period (exhaust gases flux). In a past work, the authors studied the regeneration chambers using CFD techniques assuming the checkers zone as a porous domain. The developed non-equilibrium porous model has confirmed to be a suitable tool for the design of regenerative chambers. In this work, a lower order 1-D model is presented as an additional tool to simulate the regenerative chamber during the design phase. The lower order model is tuned using the CFD 3D approach. A gases emissivity model, developed by the authors, is implemented into the 1D model in order to simulate the thermal effects of a waste gas injection in the air phase as a strategy to reduce NOx production during the following combustion process.

Keywords— Glass production plant, energy efficient regenerators, CFD, gas emissivity.

I. Introduction

In this work two numerical approaches for the investigation of regenerative chambers in glass production plants is proposed. This work is an extension of a previous paper [1] where two CFD models for regenerative chamber simulations were presented. In this paper a lower order 1-D model, calibrated using CFD, is presented as an additional simulation tool for the design of regenerative chambers with gas recirculation strategies.

The specific energy consumption of a glass furnace is about 4000 kJ every kg of glass. About 1500 kJ/kg is recovered from the hot waste gases by the combustion air using thermal regenerators. A regenerative chamber is composed of a series of refractory bricks, located inside a refractory casing, that are responsible to store heat when hot gases from combustion flow, called “hot phase”, and to release it to the combustion air in the so-called “cold phase”. The thermal regeneration process is cyclic: the exhaust gas from combustion enter from the top chamber at approximately 1400°C and exit the regenerative chamber at about 800°C; after about 20 minutes the air is fed from the bottom of the chamber to reach a temperature of about 1100°C at the top.

The energy efficiency of this process is 60-68 % against a theoretical limit value of 75-78% and this model of regenerator, conceived in 1850 by Martin-Siemens, is still the best practical solution. However, the growing concern about power consumption and pollutant emissions, requires a more strategic and efficient use of these technologies. In this prospect, a CFD approach could bring a valuable input to the design process, providing detailed analysis of consolidated solutions and support for the design of innovative concepts.

The problem of numerical modelling for regenerative heat-transfer components has been tackled by several authors for different applications [2]-[4] and the main problem is the effective modelling of the complex internal structure for CFD detailed analysis. The performance prediction of regenerative chambers is strategic to estimate the overall performance of glass furnaces. Sardeshpande [5] presents a regenerator blockage prediction model and the effect of leakage on the performance of fixed matrix regenerator was studied by Skiepko and Shah [6]; regenerator performance evaluation using numerical techniques was discussed by Foumeny and Pahlevanzadeh [7].

In a previous work [1] a CFD approach to replace the real geometry of the bricks, in the checkers zone, with a porous domain has been presented. The discretization of the real geometry of the bricks would require a too expensive mesh with an impractical CFD process to use routinely in the industrial design chain. The model must ensure a physical equivalence with the real component on these three main effects on the flow: the total pressure losses, the transfer of thermal energy (from the bricks to air and from waste gas to the bricks) and buoyancy forces (due to solid-fluid temperature gradients and chamber height).

The authors have developed a first CFD model, called “equilibrium”, for this type of regenerative chambers [8], [9] with a simplified approach for the heat flux modelling into the porous domain; the overall heat flux inside the chamber was obtained by known (estimated or from plant operation data) inlet and outlet temperature distributed linearly inside the domain. In order to be able to take into account the effect of the bricks layout and of the cruciform geometry on the heat transfer process inside the chamber, the previous model has been improved in [1] with a methodology that predicts the heat transfer inside the chamber from a given distribution of the heat transfer coefficient inside the porous domain.

This model is referred to as "non-equilibrium" model and it is an effective improvement for the regenerator simulation; it allows the effect of cruciform layout and geometry to be assessed during the design phase when the strategic decisions for the component layout are taken. The heat transfer coefficient prediction is the critical aspect for the model and it has been studied in [1] with a transient simulation performed on the real geometry of a single regenerator channel to verify the effective heat transfer inside the chambers. In [1] the results obtained with the model of non-equilibrium are compared with those obtained from the equilibrium model obtaining a good coherence between the two. In this work the development of a lower order model is presented. Using the Matlab-Simulink software, a one-dimensional finite element model of the regenerator chambers is considered. The chamber model is the base model for a more complex transient model of the whole regenerative system that is under development [10].

The lower order model is a versatile tool useful to compare various geometrical and process configurations. An algorithm specifically designed to study the thermal effects arising from the use of the waste gas recirculation system (such as that described in [9]) is presented.

The 1D model, applied to a conventional regeneration process, is compared to the CFD results. Then the model is used to simulate the thermal effects of the waste gas injection in the cold phase as a gas recirculation strategy.

II. CFD Approach for the Regenerator Chamber

As reported in [1] the Reynolds-Averaged-Navier-Stokes equations are solved with the Shear Stress Transport turbulence closure. In Fig.1 the computational domain is shown with the central domain, i.e. the checkers zone, modelled as a porous domain. As described in [8], [9] source terms in the momentum and energy equations are added to take into account friction and heat flux effects into the porous domain. The source term in the momentum equation is composed of two parts: viscous losses and inertial losses. The arrangement of the refractory bricks in the checkers zone is such as to form a kind of tube bundle with variable section, depending on the shape of the brick, with typical hydraulic diameter that varies between 150 to 200 mm. This suggests an anisotropic formulation for the porous source, equations: in facts, it identifies a preferential direction along the vertical direction of the flow in the porous domain; the flow along the other directions is inhibited multiplying by 10^2 the corresponding porous coefficients. These coefficients are obtained from a simulation campaign over a single module of bricks with its real 3D geometry by varying the mass flow rate and thermal conditions as detailed in [1]. The fundamental thermal effects inside the regenerative chamber, in addition to the porous resistance, due to heat transfer between the bricks and the fluid, need to be modelled inside the porous domain.

In the more sophisticated "non equilibrium" model the heat flux is obtained from the values of the heat transfer coefficient between fluid and solid phases and from its distribution inside the chamber. The above heat transfer coefficient is different for cold and hot cycles, depends on the brick geometry and it is estimated from a simulation campaign (as for the resistance coefficients) performed on a brick module with its actual CAD 3D geometry. The heat transfer process for cold and hot cycle is different; in the cold cycle the air flows from the bottom section to the top section into the hot regenerative chamber with strong buoyancy effects and an overall natural convection process. On the contrary, in the hot cycle, the exhaust gases flow into the chamber from the top to the bottom section with a negative pressure gradient (higher pressure for the exhaust gases in the top section from the combustion process); the forced convection heat transfer model is therefore more appropriate.

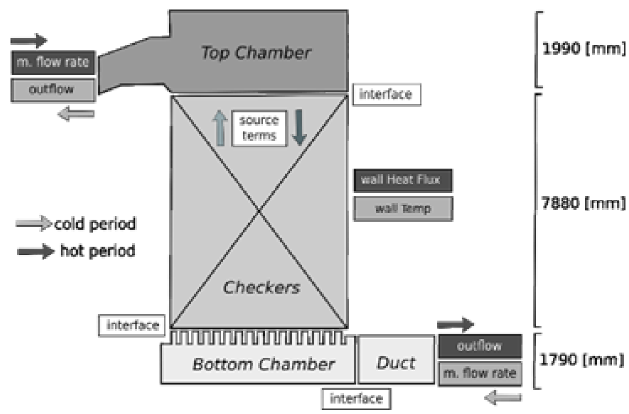

Fig.1 schematic view of the regenerator CFD model with relevant boundary conditions

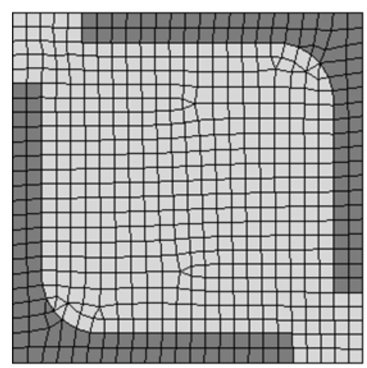

Fig. 2 mesh section of the 3D channel model with solid phase meshed (dark grey)

A. *Regenerative chambers heat transfer investigation*

The source terms in the porous model associated to the heat transfer are obtained with a simulation campaign on a model based on one channel made of two smooth refractory checkers with their real geometry.

The heat transfer analysis is performed in the channel model in order to effectively simulate the heat transfer process in the cold and hot phases taking into account the thermal inertia of the solid phase; the 3D channel model is therefore made by a the solid phase (bricks geometry) and a fluid phase as reported in Fig.2. In the sample application a five meters long channel module has been considered and discretized with an unstructured mesh of about 600 kCells. The inlet boundary temperature and flow rate (air during cold phase or gases for the hot phase) are fixed. An inlet air temperature of 420 K is fixed in the cold phase and 1410 K for the gases in the hot phase. When the hot phase is simulated, the wall temperature distribution is the one obtained from the cold phase analysis. The waste gas in the hot phase are modelled as a multi-species fluid, with the following composition: 2.3% oxygen, 11.9% carbon dioxide, 17.1% water and 68.7% nitrogen. Moreover, due to presence of water vapors and CO_2 the radiation P1 model is also considered.

Unsteady analysis are performed to reproduce the cold-hot phase change that has a typical frequency of 20 minutes. The simulation process starts with a first guessed linear temperature distribution inside the channel solid phase; the simulations are repeated by imposing the solid phase temperature from the previous analysis until convergence. When the first converged process is obtained, the cold phase (inlet air) is simulated using the solid temperature distribution from the hot phase analysis and vice versa for the hot phase (inlet gases).

Fig. 3, 5 show the distributions of the heat transfer coefficients and Fig. 6 the temperature profile of the solid for the air and waste gas cases obtained in the present sample application and used in the non-equilibrium thermal model of the regenerator.

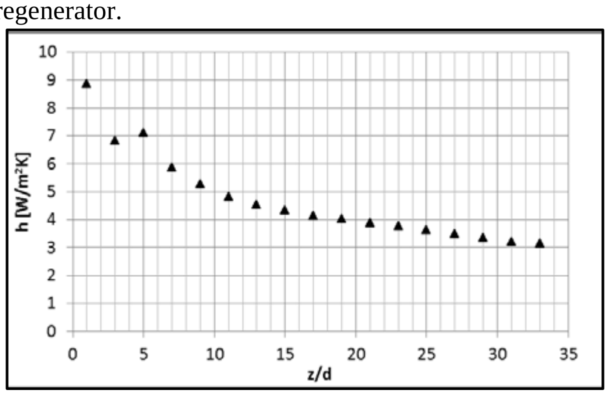

Fig.3 convective heat transfer coefficient for air along the channel

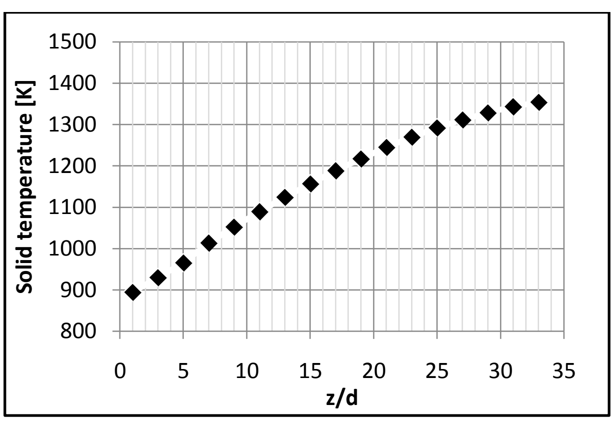

Fig.4 temperature of the solid for air phase along the channel

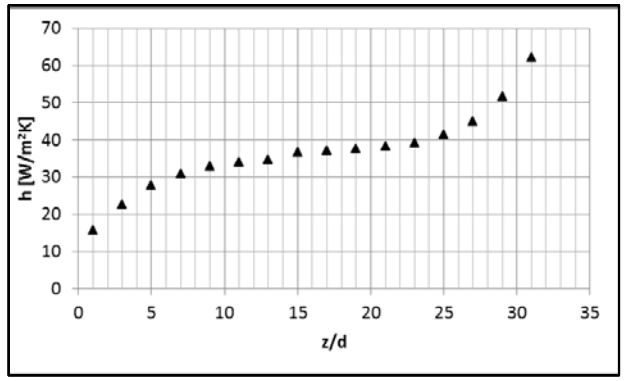

Fig.5 total heat transfer coefficient for waste gas along the channel

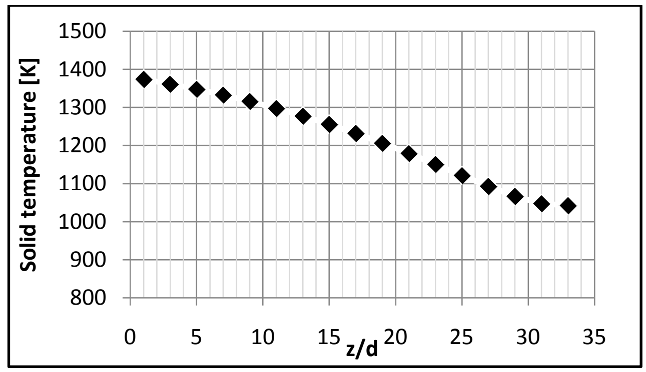

Fig.6 temperature at solid for waste gas phase along the channel

III. Lower Order Modeling of the Regenerator Chambers

A lower order model based on the 1D approach is developed in order to have a quicker solver able to predict the main effects of both chamber geometry and heat transfer (operating point) on the overall regenerator performance. The 1D model is calibrated and developed using values and assumptions from the CFD model. The 1D model is made with the finite elements method using Matlab-Simulink software.

A solid domain, representing the refractory, and a fluid domain, which may represent the cold fluid or the hot gas, constitute the regenerative chamber model. Both domains are divided into a fixed number of cells (N) along the vertical axis of the chamber, as it is show in Fig.7.

A set of equations are add to describe the heat and mass exchange between adjacent cells, using the hypothesis of no heat exchange through the casing with the external environment, no seepage or flux leakage from the casing. The fluids are treated as a mixture of ideal gases with N_2, O_2, CO_2 and H_2O. The specific heat of each substance is calculated by polynomials as a function of temperature using the same coefficients used in the CFD model.

For each fluid cell, i, an energy balance transient equation is set, as in eq. (1). The energy time variation of each fluid cell is made up by two effects: the energy transport due to the fluid mass flow and the heat transfer between the solid and the fluid.

$$m_{f,i} cv_{f,i} \frac{d\dot{T}_{f,i}}{dt} = \dot{m}_f \left(cp_{f,i-1} T_{f,i-1} - cp_{f,i} T_{f,i} \right) + \left(U_{sf,i} Sur_{sf,i} \right) \left(T_{s,i} - T_{f,i} \right) \tag{1}$$

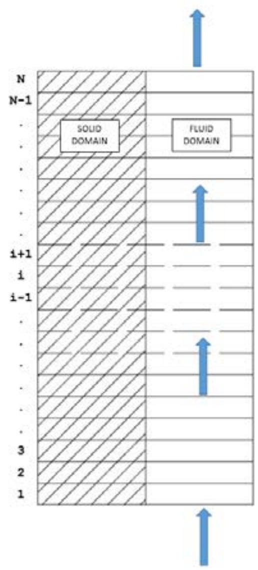

Fig.7 sketch of a single chamber domain discretization

Subscripts f and s refer to the fluid and solid domain respectively; m is the fluid mass, $\dot{m}$ its mass flow rate, cv and cp the specific heat at constant volume and pressure. The variable U_{sf} is the total thermal transmittance and Sur_{sf} is the heat transfer surface between the solid and the fluid.

For the first cell the values of $T_{f,i-1}$ and $cp_{f,i-1}$ are set equal to the fluid inlet temperature and at its specific heat. The temperature profile of the solid phase is given in input (i.e. as a linearization of the curve in Fig.4, 6).

The U_{sf} is evaluated as contribution of the thermal resistances due to the conduction trough the semi-thickness of the brick, and due to the total heat transfer coefficient, as in (2).

$$U_{sf,i} = \frac{1}{\frac{1}{h_{tot}} + \frac{th}{k_s}} \tag{2}$$

In the hot phase the total heat transfer profile is set up as an input and obtainable from the CFD model. In the cold phase, the total heat transfer coefficient is evaluated in its convective part, kept as an input from the CFD, and in its radiant part, evaluated with a specific sub-model developed for the gas emissivity. This sub-model have the aim of evaluate the thermal effects due to the introduction of waste gases in the cold phase, a technique useful for the pollutant reduction in the glass furnaces.

The radiant heat transfer coefficient, $h_{rad,i}$, is evaluate for the cold phase as in (3).

$$h_{rad,i} = \sigma\left(\varepsilon_{f,i} T_{f,i}^4 - \alpha_{f,i} T_{s,i}^4\right) \Big/ \left(T_{f,i} - T_{s,i}\right) \tag{3}$$

Where σ is the Stefan-Boltzmann constant, ε_f and α_f are the fluid radiant emissivity and absorptivity respectively.

The ε_f are evaluated with the method developed through the experimental analysis done by Hottel [11], in which the emissivity charts of CO_2 and water vapor are plotted as a function of the radiant gas pressures, the channel hydraulic diameter and the T_f; the α_f is than evaluated as function of ε_f and T_s. The emissivity graphs, available in [12] of the two chemical species are decoded in order to obtain polynomials to be implemented in the chamber model.

IV. Models Application

In [1] two CFD porous models, the equilibrium and the non-equilibrium, are compared in a conventional regenerative process, where the cold phase is made by air only and the hot phase by waste gas. In this paragraph the lower order model is firstly set-up to emulate the conventional regeneration process to compared it with the CFD non-equilibrium model and then is set-up to emulate a cold phase in which is injected a waste gas flow, as in the application of the waste gas recirculation system.

A. Non-equilibrium models comparison

The lower order model is set up with the same geometrical and boundary conditions used in [1]. The chamber is 100 m^3 with an heat transfer surface of 1650 m^2.

For the cold phase, made by air flow only, a mass flow equal to 3.77 kg/s, an inlet temperature of 420 K, and a linear regression of the heat transfer coefficient in Fig.4 are set up. The solid temperature profile is set linear from 660 K, at the flow inlet, to 1500 K at the flow outlet.

The hot phase is made by waste-gas with chemical composition reported in the section II-A, a mass flow equal to 4.88 kg/s, an inlet temperature of 1410 K, and a linear regression of the heat transfer coefficient in Fig.6 are set up. The solid temperature profile is set linear from 1375 K, at the flow inlet, to 825 K at the flow outlet.

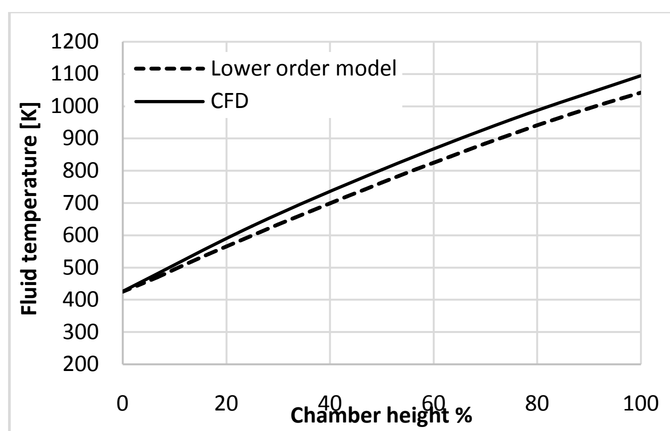

Fig.8 fluid temperature profile in the cold phase: comparison between CFD and 1D model

Fig. 8, 9 show the comparison of the fluid temperature profile calculated with the two models, for the cold and hot phase respectively. In the cold phase the lower order model has an outlet air temperature of 1040 K while the CFD model predicts 1094 K; the results differ of less than 5%. In the hot phase the outlet temperature differs only of 1°C.

The lower order model has an appreciable accuracy compared to the CFD approach and it can therefore be considered a valid tool for the thermal analysis of regenerative chambers.

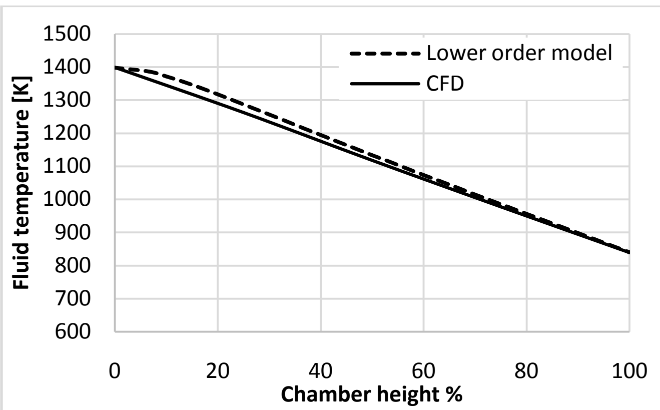

Fig.9 fluid temperature profile in the hot phase, comparison between CFD and 1D model

B. 1D model applied to the waste gas recirculation system

The lower order model has been developed also to simulate the cold phase in presence of an injection of waste gas as in a gas recirculation strategy. Below are reported the thermal analysis of the regenerator subject to different waste gas flows with the assumption of same total flow rate as in the previous case (3.77 in kg/s).

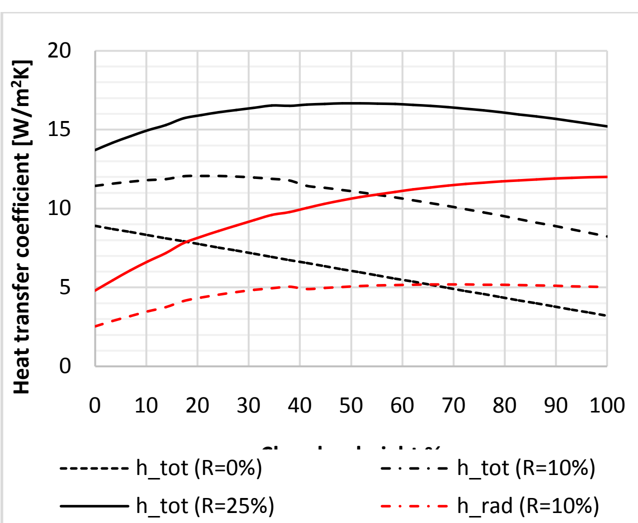

Fig.10 total and radiant heat transfer coefficients for different waste gas ratios

The waste gas mass flow is changed varying the R parameter (i.e. the ratio between the waste gas and the total mass flows). A first case is the reference with no waste gas injected ($R = 0$), the second and the third cases have R equal to 10% and 25% respectively; these are representative of two real different operative conditions of a waste gas recirculation system. The geometrical parameters of the chamber are set as in III-A, a hydraulic diameter of 140 mm is set as characteristic radiant length. The waste gas flow temperature is assumed equal to 425 K and same chemical composition reported in II-A. The temperature profile of the solid and the convective heat transfer coefficient profile are assumed linear as in III-A.

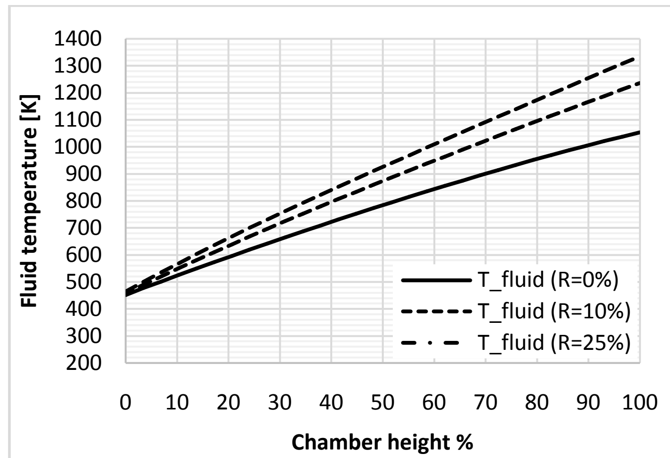

Fig.11 fluid temperature profile for different waste gas ratios

Fig.10 shows the heat transfer coefficients calculated using the gas emissivity sub-model. It is evident how the reference case is subject to convection only due to the absence of radiant gases; the total heat transfer coefficient trend is in fact linear and equal to that of the convective one. The radiative heat transfer coefficient increases with the increase of the waste gas fraction due to the higher concentration (i.e. partial pressure) of the radiant gases (CO_2, H_2O). The radiant heat transfer coefficient increases with the chamber height due to the increase of the fluid and solid temperature magnitude.

Fig. 11 shows the fluid temperature profiles by varying the waste gas ratio. The outlet temperature increases with the increase of the waste gas fraction due to the increase in radiative emission form CO_2 and H_2O.

V. Conclusion

The new lower order model has been compared with the CFD non-equilibrium model previously developed. The results from the two methods are in a very acceptable match. The lower order model can therefore be considered a useful tool for the design and analysis of regenerative chambers in addition to the more complete CFD model. Moreover, the lower order model is able to predict the thermal effects of the waste gas injection in the cold phase due to the presence of CO_2 and H_2O that have a strong effect on radiative heat transfer.

This analysis has also shown that the waste gas recirculation technique can produce desirable thermal effects (increase of the combustion mixture preheat temperature) in the regenerator in addition to the primary effect of NO_x emission reduction from the combustion process in the furnace.

ACKNOWLEDGMENT

This models and the CFD applications to glass industry presented have been developed within the PRIMEGLASS European Project (LIFE12 ENV/IT/001020) framework coordinated by Stara Glass Srl (Genova) in partnership with Università di Genova (Genova) and Stazione Sperimentale del Vetro (Venezia).

REFERENCES

[1] C. Cravero, D. Marsano, “Numerical simulation of regenerative chambers for glass production plants with a non-equilibrium heat transfer model”, Conference on Energy, Enviroment, Ecosystem and Sustainable Development, Roma , 27-29 January 2017; published on WSEAS Transactions on Heat and Mass Transfer, ISSN / E-ISSN: 1790-5044 /2224-3461, Volume 12, 2017, Art. #3, pp. 21-29;

[1] Koshelnik, A.V, “Modelling operation of system of recuperative heat exchangers for aero engine with combined use of porosity model and thermo-mechanical model”. Glass and Ceramics 2008. 65, 9-10, 301-304

[2] Zarrinehkafsh, M.T.; Sadrameli, S.M, “Simulation of fixed bed regenerative heat exchangers for flue gas heat recovery”. App Therm Eng 2004 24, 373-382

[3] Y. Reboussin, J.F. Fourmigué, JF Marthy Ph, Citti O., “A numerical approach for the study of glass furnace regenerators”, App Therm Eng 2005, 25, 2299-2320.

[4] Sardeshpande V, Anthony R, Gaitonde U.N, Banerjee R. “Performance analysis for glass furnace regenerator”, App Therm Eng 2011, 88, 4451-4458

[5] Skiepko T, Shah RK. “Modeling and effect of leakages on heat transfer performance of fixed matrix regenerators”. Int J Heat Mass Transfer 2005, 48, 1608-32

[6] Foumeny EA, Pahlevanzadeh. “Performance evaluation of thermal regenerators”. Heat Recov Syst CHP 1994; 14 (1):79-84

[7] Basso D, Briasco G, Carretta M., Cravero C, Mola A., “CFD simulation of regenerative chambers in glass industry to support the design process for thermal efficiency improvement”, Int. CAE Conference, Pacengo del Garda (Italy), 27-28 October 2014

[8] D. Basso, C. Cravero, A.P. Reverberi, B. Fabiano “CFD Analysis of Regenerative Chambers for Energy Efficiency Improvement in Glass Production Plants” Journal Energies 2015, 8(8), 8945-8961; doi:10.3390/en8088945, ISSN 1996-1073

[9] C. Cravero, A. Spoladore, “Transient numerical simulation of regenerative systems with waste gas recirculation strategies in glass production plant”. In preparation.

[10] Hottel, H. C. and Cohen, E. S. (1958), Radiant heat exchange in a gas-filled enclosure: Allowance for nonuniformity of gas temperature. AIChE J., 4: 3–14. doi:10.1002/aic.690040103

[11] Bergman, T. L., Incropera, F. P., DeWitt, D. P., & Lavine, A. S. (2011). Fundamentals of heat and mass transfer. John Wiley & Sons.