ANÁLISE DO PROCESSO DE COMBUSTÃO DE CARVÃO, DO ESCOAMENTO E DA TRANSFERÊNCIA DE CALOR NUM GERADOR DE VAPOR DE UMA CENTRAL TERMELÉTRICA USANDO CFD

CFD analysis of the combustion, gas flow, and heat exchange processes in a boiler of a thermal power plant

SILVA, C.V., INDRUSIAK, M.L.S. and BESKOW, A.

RESUMO: O papel estratégico da geração de energia e os atuais danos causados ao ambiente relacionados ao efeito estufa, a eficiência energética e exergética da queima de combustíveis fósseis apontam para a importância dos estudos de complexos processos físicos e químicos que ocorrem no interior de caldeiras de grandes centrais térmicas. O estado da arte em dinâmica dos fluidos computacional e a disponibilidade comercial de códigos numéricos incentivam cada vez mais os estudos dos processos de combustão. No presente trabalho o software comercial CFX@Ansys Europe Ltd. é utilizado para estudar o processo de combustão, os escoamento dos gases e a transferência de calor num gerador de vapor de uma usina termelétrica de 160 MW abastecida com carvão pulverizado. O comportamento dos escoamentos de ar e de carvão pulverizado através dos queimadores foi analisado através de simulação numérica, assim como o escoamento tridimensional dos gases de exaustão através da câmara de combustão, incluindo os trocadores de calor. Foi verificado que o código computacional apresenta uma boa sensibilidade à variações das condições de contorno e de entrada do problema, principalmente com relação a formação de NOx.

Palavras-chave: Combustão, Dinâmica dos Fluidos Computacional, Plantas de Potência.

ABSTRACT: The strategic role of energy and the current concerning with greenhouse effects, energetic and exergetic efficiency of fossil fuel combustion greatly enhance the importance of the studies of complex physical and chemical processes occurring inside boilers of thermal power plants. The state of the art in computational fluid dynamics and the availability of commercial codes encourage numeric studies of the combustion processes. In the present work the commercial software CFX@Ansys Europe Ltd. has been used to study the combustion processes, the flue gas e heat transfer into the boiler of a 160 MW thermal power plant. The behavior of the air and pulverized coal flows through the burners was analyzed and the three-dimensional flue (or exhaust) gas flow through combustion chamber and heat exchangers was reproduced in the numeric simulation. It was verified that the code shows a good sensibility to variations in inlet and boundary conditions, mainly to respect to NOx formation.

Keywords: Combustion, Computational Fluid Dynamic, Thermal Power Plant.

Introduction

In some parts of the world coal is an important energy resource for meeting the future demand for electricity, as coal reserves are much greater than those of other fossil fuels. However, the efficiency and clean utilization of this fuel is a major problem in combustion processes. In recent years, the interest on performance optimization of large utility boilers has become very relevant, aiming at extending their lifetime, increasing the thermal efficiency and reducing the pollutant emissions, particularly the NOx emissions. Combustion comprises phenomena such as turbulence, radiative and convective heat transfer, particle transport and chemical reactions. The study of these coupled phenomena is a challenging issue. The state of the art in computational fluid dynamics and the availability of commercial codes encourage numeric studies of the combustion processes. In the present work a commercial software, CFX©Ansys Europe Ltd., has been used to study the coal combustion process in a 160 MW commercially operated thermal power plant, with the objective of simulate the operation conditions and identify inefficiency factors.

Coal reserves in Brazil, which are used mainly for electricity production, are enough to meet the next 200 years demand. Nonetheless, in order to face the competition from renewable, natural gas and nuclear energy sources, some main problems must be solved, as to reduce CO_2 emissions through increasing efficiency (WILLIAMS ET AL., 2000). Also NO_x and SO_x emissions should be reduced to environmentally acceptable levels. At this way, an efficient operation of combustion chambers of these plants depends on the proper knowledge of the oxidation reactions and heat transfer from the combustion products to the chamber walls and heat exchangers, which requires a detailed analysis of the governing mechanisms.

Li et al. (2003) numerically investigated the combustion process using a pure two-fluid model (instead of the Eulerian gas - Lagrangian particle models) for simulating tree-dimensional turbulent reactive flows and coal combustion. To improve the simulation of the flow field and NOx formation, a modified $k-\varepsilon-k_p$ two-phase turbulence model and a second-order-moment (SOM) reactive rate model are proposed. The proposed models are used to simulate NO formation of methane-air combustion, and the prediction results are compared with those using the pure presumed-PDF (Probability Density Function)-finite-reaction-rate model and experimental data. The proposed models are also used to predict the coal combustion and NOx formation at the exit of a double air register swirl pulverized-coal burner. The predicted results indicate that a pulverized coal concentrator installed in the primary air tube of burner has a strong effect on the coal combustion and NOx formation.

In a numerical investigation, Kurose et al. (KUROSE; MAKINO end SUZUKI, 2004) employed a tree-dimensional simulation to the pulverized coal combustion field in a furnace equipped whit a low-NOx burner, called CI-α, to investigate in details the combustion processes. The validities of existing NOx formation and reduction models were investigated too. The results show that a recirculation flow is formed in high-gas-temperature region near the CI-α burner outlet, and this lengthens the residence time of coal particles in this high-gas-temperature region, promotes the evolution of volatile matter and the process of char reaction, and produces an extremely low-O_2 region for effective NO reduction.

Zhang et al. (2005) present a numerical investigation on the coal combustion process using an algebraic unified second-order moment (AUSM) turbulence-chemistry model to calculate the effect of particle temperature fluctuation on char combustion. The AUSM model was used to simulate gas-particles flows, in coal combustion including the sub-models as the $k-\varepsilon-k_p$ two-phase turbulence model, the EBU-Arrhenius volatile and CO combustion model, and the six-flux radiation model. The simulation results indicate that the AUSM char combustion model is far preferable to the old char combustion model, since the later totally eliminates the influence of particle temperature fluctuation on char combustion rate.

Bosoaga et al. (2006) presents a study developing a CFD model for the combustion of low-grade lignite and to characterize the combustion process in the test furnace, including the influence of the geometry of burner and furnace. A number of computations were made in order to predict the effect of coal particle size, the moisture content of lignite, and the influence of combustion temperature and operation of the support methane flame on the furnace performance and emissions. The influence of lignite pre-drying was also modeled to investigate the effects of reduced fuel consumption and CO_2 emissions. It was found that the increase of moisture tends to reduce NOx, and the methane support flame greatly increased NOx.

In another work, Backreedy et al. (2006) present a numerical and experimental investigation on the coal combustion process to predict the combustion process of pulverized coal in a 1 MW test furnace. The furnace contains a triple-staged low-NOx swirl burner. A number of simulations were made using various coals in order to calculate NOx and the unburned carbon-in-ash, the later being a sensitive test for the accuracy of the char combustion model. The NOx incorporates fuel-NO, thermal, and prompt mechanisms to predict the NO formation on the combustion processes.

Kumar and Sahur (2007) study the effect of the tilt angle of the burners in a tangentially fired 210 MW boiler, using commercial code FLUENT. They show the influence of the tilt angle in the residence time of the coal particles and consequently in the temperature profiles along the boiler.

Asotani et al. (2008) also using the code FLUENT, study the ignition behavior of pulverized coal clouds in a 40 MW commercial tangentially fired boiler. The results for unburned carbon in ash and for outlet temperature were validated respectively by the operating data and by the design parameter. A qualitative comparison between the results for temperature and ignition behavior in the vicinity of the burners was made, using the images of a high temperature resistant video camera system.

The strategic role of energy and the current concerning with greenhouse effects enhance the importance of the studies of complex physical and chemical processes occurring inside boilers of thermal power plants. Combustion comprises phenomena such as turbulence, radiative and convective heat transfer, particle transport and chemical reactions. The study of these coupled phenomena is a challenging issue. The state of the art in computational fluid dynamics and the availability of commercial codes encourage numeric studies of the combustion processes.

In the present work a commercial CFD code, CFX@Ansys Europe Ltd., was used to study the coal combustion process in a 160 MW thermal power plant erected in the core of the Brazilian coal reserves region, with the objective to verify the specific operation conditions to respect to NOx formation.

Background

The set of equations solved by CFX are the mass, momentum to incompressible flow, energy and chemical species conservation equations and the equations of state of real gas. An Eulerian description is adopted for the fluid phase and a Lagrangean tracking model for the coal particles. The κ-ω turbulence production–dissipation model is applied to solve the closure problem of the averaged Navier Stokes equations (CFX SOLVER THEORY, 2004).

CFX calculates coal combustion by combining a particle transport calculation of the coal particles with a Arrhenius-Eddy Dissipation model to calculate the combustion of the volatile gases (assuming methane and carbon monoxide as devolatilization products), also using two global steps to calculate its oxidation. The combustion of a coal particle is a two stage process: the coal devolatilization followed by the oxidation of the residual char to leave incombustible ash. Arrhenius equations are used to predict the devolatilization process and the Field Model (CFX SOLVER THEORY, 2004) is used to predict the char oxidation. Devolatilization was usually modeled by two competing reactions in order to deal with the strong dependence on temperature and heating rate of the bituminous coal (CFX SOLVER THEORY, 2004). The two equations have different rate parameters and volatile yields. The yield fractions for the lower temperature equation were obtained from proximate analysis and to the ones for the higher temperature equation were given the values suggested by Li et al. (2003).

The complete chemical reaction of the raw coal used at this work, including two devolatilisation processes, is modeled according to the basic scheme showed in Fig. 1. As basic assumptions, it is considered that the mass fractions of volatiles are 0.3636 of methane and 0.6364 of carbon monoxide, and that the combustion processes of these volatiles occur at finite rates. The methane oxidation is modeled by two global steps, given by:

$$2CH_4^{(16)} + 3\left(O_2^{(32)} + 3.76N_2^{(28)}\right) \rightarrow 2CO^{(28)} + 4H_2O^{(18)} + 11.28N_2^{(28)} \tag{1}$$

$$2CO^{(28)} + 1\left(O_2^{(32)} + 3.76N_2^{(28)}\right) \rightarrow 2CO_2^{(28)} + 3.76N_2^{(28)} \tag{2}$$

Equation (2) also models the combustion of carbon monoxide resulting from the devolatilisation processes.

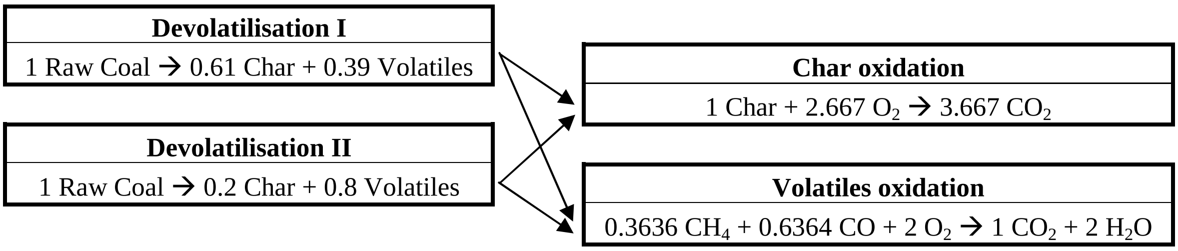

Figure 1 - Basic scheme of the full chemical reactions of the raw coal.

The model adopted for the char burnout computes the rate of the reaction taking into account the rate of diffusion of oxygen and its partial pressure at the particle surface (KANURY, 1975). Particle size plays an important role in that mechanism and was modeled by a Rosin-Rammler statistical distribution (BROWN, 1975), with the parameters adjusted from pulverized coal analysis.

To predict the NOx formation the Zeldovich model (thermal-NO) is used, along with the Fennimore model (prompt-NO), where the first, predominant at temperatures above 1800 K, is given by tree-step chemical reaction mechanisms (CFX SOLVER THEORY, 2004):

$$O^{(16)} + N_2^{(28)} \rightarrow NO^{(30)} + N^{(14)} \tag{3}$$

$$N^{(14)} + O_2^{(32)} \rightarrow NO^{(30)} + O^{(16)} \tag{4}$$

In sub or near stoichiometric conditions, a third reaction is also used

$$OH^{(17)} + N^{(14)} \rightarrow NO^{(30)} + H^{(1)} \tag{5}$$

where the chemical reaction rates are predicted by Arrhenius equation.

The Discrete Transfer Radiation Model - DTRM is used to predict the radiation heat transfer of the gases to the walls. A gray gas model is adopted and its gray gas coefficient was set to $0.6 m^{-1}$.

The heat transfer across boundaries and in heat exchangers is also considered. The combustion processes occurring in the boiler generate a huge amount of thermal energy which is transferred to the working fluid (water) in the heat exchangers by means of two basic mechanisms: convection and thermal radiation. In fact, heat transfer to the walls in a utility boiler is mainly due to radiation and the convective heat transfer has only a minor contribution (XU, AZEVEDO e CARVALHO, 2000). Conversely, heat transfer in the tube banks, which were presented as porous media, was modeled by means of volumetric sink coefficients representing the total amount of thermal energy transferred to working fluid inside the tubes of each bank. The pressure losses due to the tube banks are also modeled assigning quadratic directional loss coefficients to the porous media, computed from the tube bank geometry data (KNUDSEN, 1958).

More detailed of formulation can be found at Silva et al. (2007) and CFX Solver Theory (2004).

Geometry and Mesh Settings, Boundary Conditions and Convergence Criteria

The boiler under consideration is part of a pulverized coal thermal power plant. The combustion chamber modeled is rectangular in shape with four burners firing from each corner, producing a large vortex in the center of the chamber. The evaporation process occurs mainly in the tubes covering the boiler walls. In the upper middle of the boiler are the reheater (LTR, HTR), superheater (LTS, HTS) and economizer (ECO2) tube banks. The second stage of the boiler comprises a large rectangular curved duct, the first economizer (ECO1) tube bank and the regenerative air heater (Ljungström), Fig 2-a. From there the flue gases are directed through the electrostatic precipitator to the chimney. The entrance to the second stage was considered the outlet of the domain. The primary and secondary air paths are also shown, departing from the fans (PAF, SAF) through the air heater and coal silos (only for the primary air) to the burners. The burner disposition at the corners is shown at Fig 2-b.

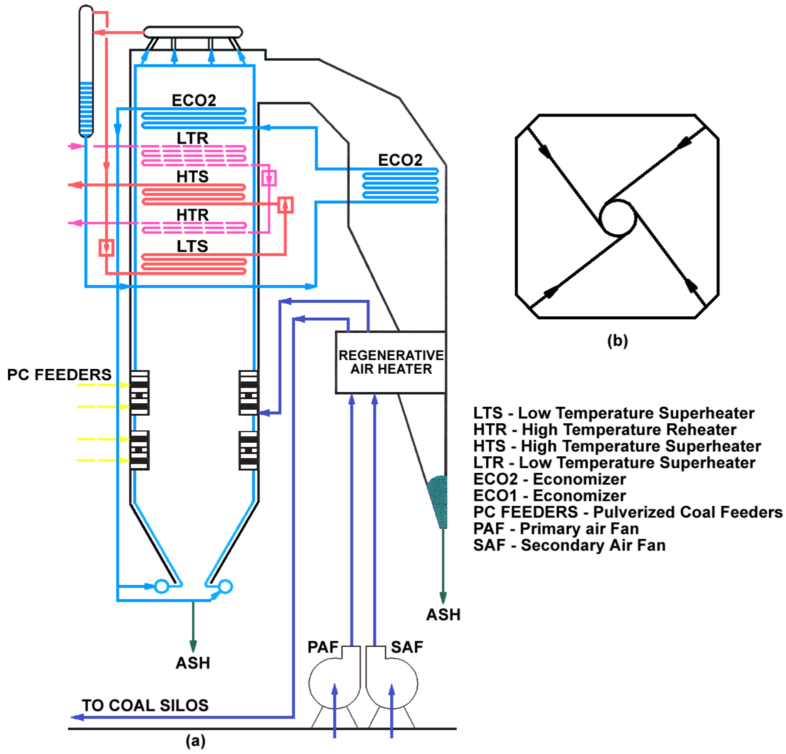

Figure 2 – (a) General disposition of the boiler components; (b) Horizontal cross section.

The discretization of the 3D geometry was done using tetrahedral volumes. At the walls prismatic volumes were used in order to capture the boundary layer behavior. The mesh used is static and it has approximately 1.6×10^{6} elements.

The boundary conditions were obtained from the design data set and also from the operation data sheets. The operating conditions considered were the rated ones, for 160 MW. The following parameters were considered:

The initial condition considered was the boiler completely full with air at normal conditions of temperature and pressure, with no flow.

The convergence criterion adopted was the RMS of the residual values less than 1.5×10^{-6}, and the set of equations is solved to the steady state.

The models that comprises the combustion modeling are implemented in sequence until the completion of the coal combustion processes, all of them obeying the same convergence criterion: 1– Isothermal air flow using 3-D Navier-Stokes equations with RANS – Reynolds Average Navier-Sokes and a turbulence model; 2– Buoyancy due to gravitational forces; 3–

Thermal energy conservation equations; 4–chemical species conservation equations with a gas fuel, e. g. methane and carbon monoxide in two global steps; 5– Combustion model based on Eddy-Break-up model to establish an initial temperature field; 6–Model of chemical kinetics, as Arrhenius model; 7– Departing from the fields already obtained, replacement of the methane by the actual fuel, coal, with the devolatilisation model and char combustion model, considering uniform particle size; 8– Radiation model, here the DTRM; 9– Model of distribution of particle size; 10– Model for NOx formation. The whole implementation of the formulation spent approximately 60 days, with 24 hours/day of computation.

The inlet, outlet and contour conditions were set as follows:

Inlet: The inlet conditions are the design ones for air and coal flows entering the domain from the burner nozzles. Total primary and secondary combustion air and pulverized coal mass flow rates were set as 79.5 kg/s, 100 kg/s and 36 kg/s respectively. Temperatures of primary air and coal, and secondary combustion air were set as 542 K and 600 K respectively. Pulverized coal size was modeled by a probabilistic distribution (Rosin-Rammler) and limited between 50 μm and 200 μm.

Outlet: The outlet boundary is the flue gas passage to the second stage, where the mean static pressure was set.

Boiler walls: The boiler walls are covered with slanting tubes from the bottom until the beginning of the heat exchangers region; from there to the top the tubes are vertically positioned. Wall roughness, temperature and thermal radiation coefficients were set for that two wall regions.

The reheater (LTR, HTR), superheater (LTS, HTS) and economizer (ECO2) tube banks are modeled as porous media, and an energy sink coefficient was set in order to emulate the pressure losses.

Results

The temperature field is shown at Fig. 3-a (right) for a vertical plane diagonally positioned. The large amount of heat released by the devolatilization and oxidation of the volatiles is pointed up by the near red regions at the edge of the flames originated at each burner. Devolatilization is the first reaction of the combustion process and takes place where the air and coal mixture injected by the burners achieved the adequate temperature. The central vortex created by the tangential layout of the burners is visible at the center of the combustion chamber.

As the flow moves to the outlet the heat is exchanged with the walls and tube banks, creating the temperature gradient shown in the figure. The temperature and velocity fields are presented in a superimposed way at Fig. 3-g to 3-k for horizontal planes corresponding to the four burner levels and a level just upstream the burner region. The temperature color scale is the same for all the figures. At the lower burner levels the general temperature distribution show lower values than at the higher levels. At Fig. 3-g the temperature presents a trend to equalization, due to both the absence of new inflows and the strong turbulence and vorticity of the flow. The velocity fields, represented by means of vectors, show that at the lower burner level the vortex region is narrow and increases in the upstream direction, due to the vorticity moment imparted by the burners jets at each level. Figure 3-g shows the final aspect of the vortex which dominates the section, with a characteristic dimension of the same magnitude of the boiler wall horizontal length.

There is an intense formation of volatiles very near to the burner nozzles, denoting the action of the first reaction which is activated at relatively low temperatures. The oxidation of the resulting volatile yields is almost immediate, according to the set of equations which models the combustion process.

Figures 3-b to 3-f show the distribution of NO_x mass fraction along the boiler. The NO_x formation takes place mainly after the coal devolatilization and volatile oxidation, at the top edges of the air-fuel jets from each burner, where the higher temperatures were achieved. The major role of high temperature along with high oxygen concentration levels in NO_x formation is also emphasized by the impressive enlargement of NO_x production at the higher burner levels. An enhancement of the oxygen concentration is expected at these levels, where the inlet air jets are reinforced by the residual oxygen of the lower levels.

The results obtained were analyzed and compared with known data of the boiler operation. The main control parameters used to validate the results shown at Tab. I were the heat transfer rate at the walls, the outlet temperature and the mass fractions of gases at the boiler outlet where there are regular measurements. The experimental data presented at Tab I are obtained from the plant operation control system. The simulation results for heat rate, outlet temperature and %O_2 match quite well with experimental data. The additional amount of O_2 and CO in experimental results point out that the actual combustion process is less efficient than at the simulation and produces more CO and O_2 and less CO_2. Indeed, the maintenance staff information is that there is unburned coal at the ash. Several simulations were done with more and less fuel and air and the results indicate that the model response is adequate to those variations. However more experimental information is necessary in order to improve the agreement between real data and simulation results.

The NO_x results do not match at all. This is an expected result, because only prompt and thermal NO were simulated and the fuel NO, which accounts for 75-95% of the total NO in coal combustors (KUROSE, MAKINO AND SUZUKI, 2004) was not simulated. This is the next goal of the research.

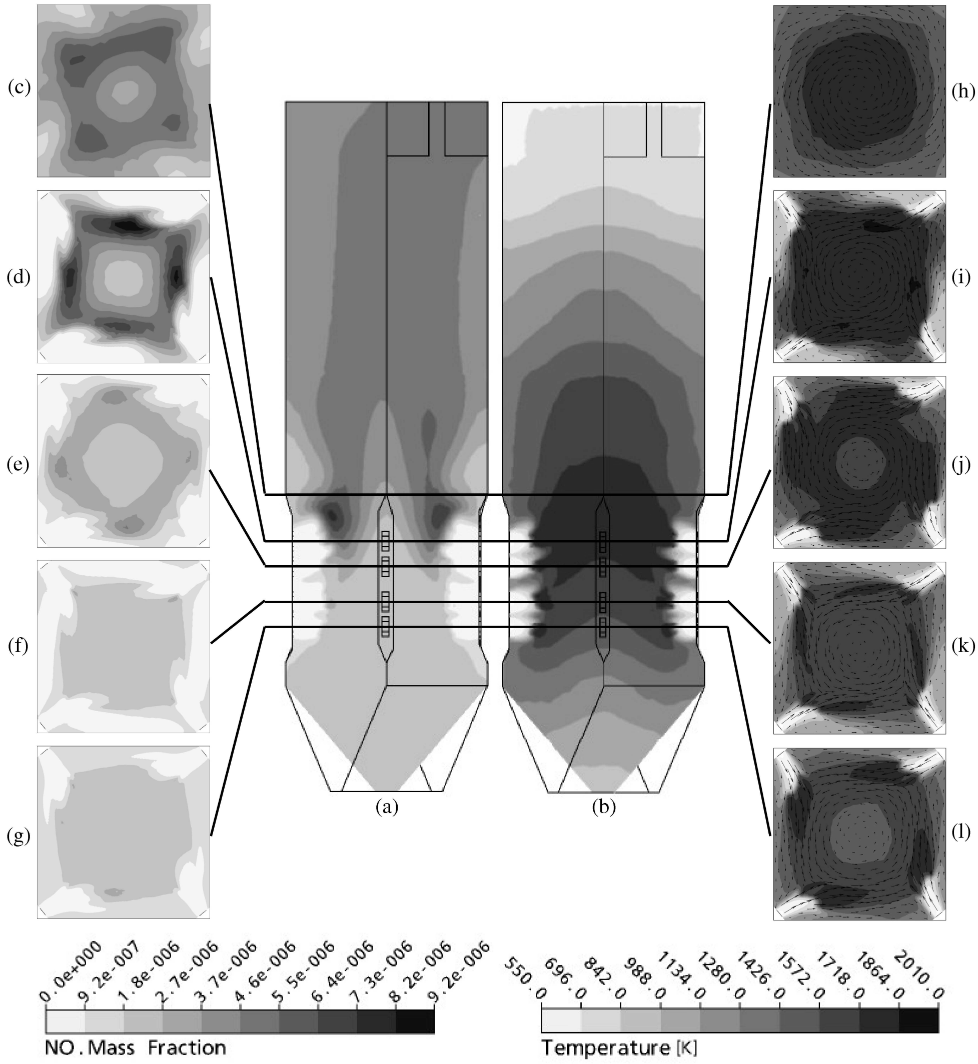

Figure 3: (a) NO_x mass fraction and (b) temperature fields in the boiler. (c) to (g): NO_x mass fraction field in the boiler, at horizontal planes at positions indicated by the lines. (h) to (i): Temperature field at the same horizontal planes. The superimposed vectors represent the velocity field in the planes.

Table I - Main control parameters used to validate the results.

	Heat transfer rate [kW/m²]	**Outlet temperature [ºC]**	**% O_2**	**% CO_2**	**CO [ppm]**	**NO_x [ppm]**
Experimental data	170	414	6.5	13.2	58	168
Simulation results	177	484	4.4	20.8	0.7	8.53

Conclusions

The general description of the numeric model of a thermal power plant boiler using a commercial CFD code was presented in this article. The aim of the work is the use of the results to better understand the complex processes occurring within the boiler. Some results were presented and discussed. The temperature and velocity fields are in agreement with the expected behavior of a coal combustion chamber.

The code shows a good sensibility to variations in inlet and boundary conditions and this was explored in order to study the performance of the boiler at out of design and part-load operation conditions. Also the combustion, heat exchange processes and NO_x formation responses to other conditions at the burners, like the vertical tilt, were studied.

AUTORES

Cristiano Vitorino da Silva[1] – Doutor em Engenharia Mecânica pela Universidade Federal do Rio Grande do Sul - UFRGS, atualmente é professor e coordenador do Curso de Engenharia Industrial Mecânica da Universidade Regional Integrada do Alto Uruguai e das Missões – URI – Campus de Erechim.

Maria Luisa Sperb Indrusiak – Doutora em Engenharia Mecânica pela Universidade Federal do Rio Grande do Sul - UFRGS, atualmente é pesquisadora e consultora da SI Consultoria e Pesquisa - RS.

Arthur Beskow – Mestre em Ciências pela Pontifícia Universidade Católica do Rio Grande do Sul – PUC-RS, atualmente é pesquisador do LABSIM – Laboratório de Simulação Numérica do Curso de Engenharia Química da Universidade Federal do Rio Grande do Sul - UFRGS.

1- Correspondence to: cristiano@uricer.edu.br; Campus II – RS 331, nº345, Bairro Demoliner, Erechim, RS.

REFERENCES

WILLIAMS, A.; POURKASHANIAN, M.; JONES, J. M.; SKORUPSKA, N.. **Combustion and Gasification of Coal**, *Taylor & Francis*, New York, 2000.

LI, Z. Q.; WEI, Y.; JIN, Y., 2003. **Numerical simulation of pulverized coal combustion and NO formation**, *Chemical Engineering Science*, **58**, 5161.

KUROSE, R.; MAKINO, H. AND SUZUKI, A., 2004. **Numerical analysis of pulverized coal combustion characteristics using advanced low-NOx burner**, *Fuel*, **83**, 693.

ZHANG, Y.; WEI, X., ZHOU; L. AND SHENG, H., 2005. **Simulation of coal combustion by AUSM turbulence-chemistry char combustion model and a full two-fluid model**, *Fuel*, **84**, 1798.

BOSOAGA, A.; PANOIU, N.; MIHAESCU, L.; BACKREEDY, R.I., MA, L.; POURKASHANIAN, M. AND WILLIAMS, A., 2006. **The combustion of pulverized low grade lignite**, *Fuel*, **85**, 1591.

BACKREEDY, R.I.; FLETCHER, L.M.; MA, L., POURKASHANIAN, M. AND WILLIAMS, A., 2006. **Modelling Pulverised Coal Combustion Using a Detailed Coal Combustion Model**, Combust. Sci. and Tech., **178**, 763.

KUMAR, M. AND SAHU, S. G., 2007. **Study on the effect of the Operating Condition on a Pulverized Coal-Fired Furnace Using Computational Fluid Dynamics Commercial Code**, *Energy & Fuels*, **21**, 3189.

ASOTANI, T., YAMASHITA, T., TOMINAGA, H., UESUGI, Y., ITAYA, Y. AND MORI, S., 2008. **Prediction of ignition behavior in a tangentially fired pulverized coal boiler using CFD**, *Fuel*, **87**, 482.

CFX **Solver Theory**, 2004.

KANURY, A. M.. **Introduction to Combustion Phenomena**, *Gordon and Beach Science Publishers*, New York, 1975.

BROWN, W. K., 1995. **Derivation of the Weibull distribution based on physical principles and its connection to the Rosin-Rammler and lognormal distributions**, *Journal of Applied Physics*, **78**, 2758.

XU, M.; AZEVEDO, J. L. T. AND CARVALHO, M. G., 2000. **Modelling of the combustion process and NO_x emission in a utility boiler,** *Fuel*, **79**, 1611.

KNUDSEN, J. G.. **Fluid Dynamics and Heat Transfer**, *Mc Graw Hill*, 1958.

SILVA, C. V.; INDRUSIAK, M. L. S.; BESKOW, A. AND KAEHLER, J.W.M., 2007. **CFD analysis of the pulverized coal combustion processes in a boiler of a thermal power plant: influence of tilted inlet flow of fuel and air on the NOx formation.** In: *19th International Congress of Mechanical Engineering*, ABCM , November 5-9, Brasília, DF.