Thermal Power Plant Boiler Misoperation - Case Study Using CFD

Indrusiak, M. L. S. [*, 1]; Beskow A. B.[2]; da Silva, C. V.[2]

[1]UNISINOS – Universidade do Vale do Rio dos Sinos; São Leopoldo – RS, Brasil.
[2]URI – Universidade Regional Integrada do Alto Uruguai e das Missões; Erechim – RS, Brasil.

Abstract
The actual operation condition of a thermal power plant, where the water seal in the bottom of the boiler presented an air leakage, was simulated using the commercial CFD code CFX. The results were compared with the ones for the hypothetic situation where the leakage does not exist and the total amount of air required for the combustion would be furnished by the secondary air feeds. The misoperation condition affects the performance and the flow dynamics of the boiler and also the NO_x formation.

Introduction

A substantial portion of the electric energy generated worldwide comes from fossil fuel sources. In addition to economic considerations, the current concerning with greenhouse effects enhances the importance of the efforts done for a proper and efficient thermal power plant operation.

Coal 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, as Backreedy et al., 2006, and Kumar and Sahur, 2007. In the present work the commercial code CFX 10.0 has been used to study the coal combustion process in a boiler of a 160 MW commercially operated thermal power plant, with the objective of simulate the operation conditions.

The purpose of the study is the analysis of an actual operation condition where the water seal in the bottom of the boiler presented an air leakage. The amount of air leak was evaluated by the plant staff and also computed by stoichiometric balance. The results were compared with the ones for the hypothetic situation where the leakage does not exist and the total amount of air would be furnished by the secondary air feeds.

Background

The set of equations solved by CFX are the mass, the momentum, the energy and the 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 k-ω turbulence production–dissipation model is applied to solve the closure problem of the averaged Navier Stokes equations.

CFX calculates coal combustion by combining a particle transport calculation of the coal particles with a combined finite rate Arrhenius-Eddy Dissipation model to calculate the combustion of the volatile gases (assuming just methane and carbon monoxide as devolatilization products), using yet two global steps to calculate the methane 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 is used to predict the char oxidation. Devolatilization was usually modeled with two competing reactions in order to deal with the strong dependence on temperature and heating rate of the bituminous coal. 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 model adopted for the char burn out 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, 1995, with the parameters adjusted from pulverized coal analysis data.

To predict the NOx formation the Zeldovich model is used, with two different mechanisms, the thermal-NO and the Prompt-NO. The DTRM – Discrete Transfer Radiation Model is used to predict the radiation heat transfer of the gases to the walls. A gray gas model is adopted.

In addition to the gas and particles flow and the heat released by the chemical reactions, the heat transfer across boundaries and in heat exchangers was 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 et al., 2000). Conversely, heat transfer in the tube banks, which were simulated as porous media, was modeled by means of volumetric sink coefficients representing the total amount of thermal

[*] Corresponding author: mlsperb@unisinos.br
European Combustion Meeting 2009.

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).

Geometry, settings and convergence criteria

The boiler under consideration is part of a pulverized coal power plant operating in a subcritical steam cycle. The combustion chamber 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, superheater and economizer tube banks. The second stage of the boiler comprises a large rectangular curved duct, the first economizer tube bank and the regenerative air heater. From there the flue gases are directed through the electrostatic precipitator to the chimney. See de Fig. 1.

The domain considered comprises the first stage of the boiler: the combustion chamber with the burners at the corners and the heat exchangers until the top. The entrance to the second stage was considered the outlet of the domain. The discretization of the geometry was done using tetrahedral volumes. As the boiler height corresponds to only six equivalent diameters of the boiler, the boundary layer is not developed at the whole domain. Nevertheless, at the walls prismatic volumes were used in order to capture the boundary layer behavior. The mesh used has approximately 1.5×10^6 elements of unequal size, with about 90% of them in the reactive region near the burners.

The convergence criterion adopted was the RMS – root mean square of the residual values. Values less than 1.5×10^{-6} were achieved at all equations. The convergence was achieved only by the gradual implementation of the models. Results for the design data set are presented at Indrusiak et al. 2007 and Silva et al. 2008.

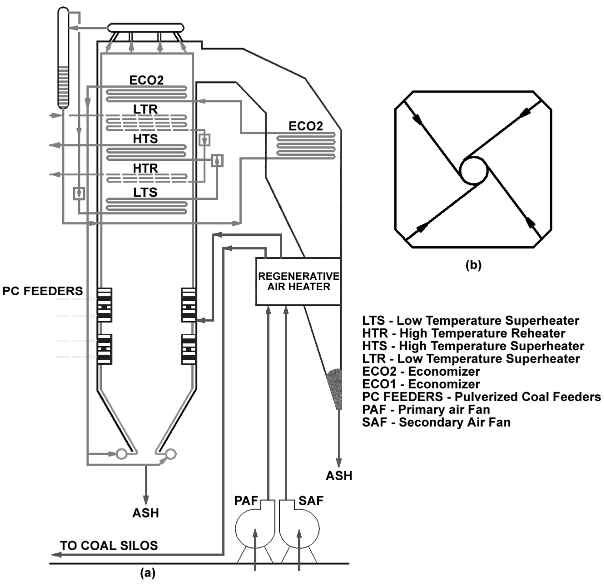


Fig. 1: (a)Boiler details. (b)Horizontal cross section.

Boundary Conditions

The boundary conditions were obtained from the operation data sheets. The operating conditions considered were the actual ones, for 160 MW. The main values are presented at Tab. 1. The following parameters were considered:

Inlet: The inlet conditions are those for air and coal flows entering the domain from the burner nozzles. Primary and secondary combustion air and pulverized coal mass flow rates and temperatures and also pulverized coal size probabilistic distribution parameters were set. Atmospheric air is considered. The raw coal composition was adjusted according to the proximate analysis. The amount of air entering the water seal was evaluated by the plant staff and assigned to the bottom inlet boundary. For comparison, the same air quantity was assigned to the secondary air inlet in another simulation.

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.

Table 1: Boundary conditions values

	A	B	C
Primary air temperature (ºC)	310	310	310
Secondary air temperature(ºC)	293	293	293
Primary air flow (kg/s)	30	30	30
Secondary air flow (kg/s)	48	48	62
Coal flow (kg/s)	22	22	22
Water seal flow (kg/s)	30	70	0
Water seal air temperature(ºC)	40	40	-
Outlet pressure (Pa)	-116	-116	-116

Results

The analysis of the flow behavior, temperature field and heat exchange at the walls was done for the three situations:

A- with the value of bottom air leakage evaluated by the power plant staff;

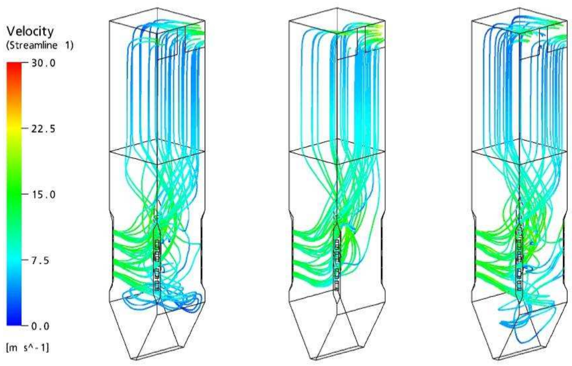
B- with the value of total amount of air, computed by stoichiometric balance, minus the primary and secondary air, assigned to the bottom leakage;

C- with the value assigned to the air leakage in case A now added to the secondary air.

Case-A represents exactly the data furnished by the power plant staff. Case-B try to represent a more realistic state, nevertheless it should be considered that there are other minor air leaks distributed along the boiler and the flow assigned to the bottom seal could be a bit over evaluated.

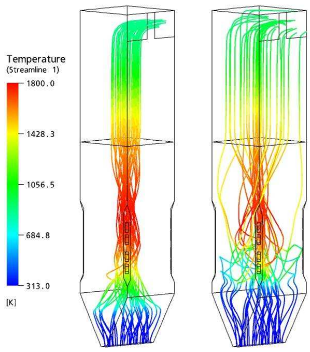
The data furnished by the plant staff are also the basis of Case-C, which distinguishes from Case-A only by the change (move) to the secondary air feed of the flow originally assigned to the bottom seal.

The cold air entering at the bottom creates a downside-up stream which flows mainly along the vertical axis of the boiler. Figure 2 shows, for cases A and B, the flow streamlines of cold air and Fig. 3 shows the perturbation of the flow from the burners due to the action of the cold air flow. Despite of the 15 ^{o} downside tilt of the burners, the reacting flow vortice, in cases A and B evolve directly to the top, thus diminishing the residence time of the coal particles in the reactive zone.



A B C

Figure 2: Streamlines of the flow departing from one corner burners, for cases A, B and C

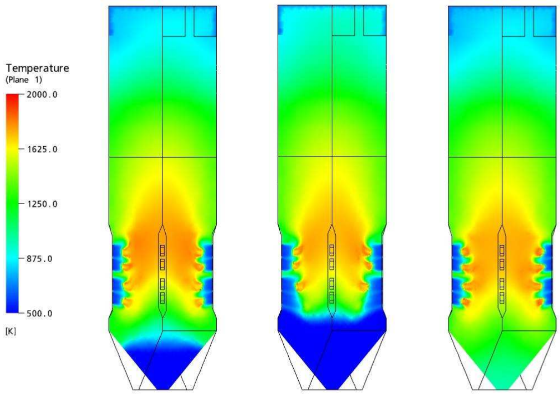


A B

Figure 3: Streamlines for the flow coming from the leakage at the water seal, cases A and B.

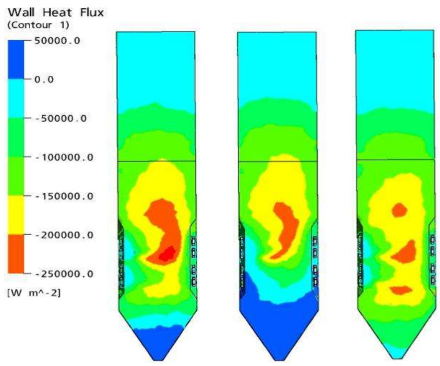
The leak air quantity is large enough to reduce the temperature at the boiler lowest part. Figure 4 shows the temperature distribution at the vertical plane positioned diagonally (from one corner with the burners to the opposite one). The cold air entering at the bottom at cases A and B creates a colder region which displaces downside-up the reactive zone. For case B the cooling of the bottom is more impressive, as more cold air is imparted through the water seal.

Figure 5 shows the total amount of heat flux (convective and radiative transfer) through the boiler walls. There are three main regions of transference: The reactive region, where the temperature are higher and a large amount of heat is transferred to the water and vapor at the wall tubes, mainly by radiaction; an upper region, where the main mechanism of transference is convection at tube banks and in much lesser extent in the walls; and a third region, at the bottom. For cases A and B this region is characterized by an inversion of heat flux at the walls, because the leakage at the bottom seal made the temperature at the region lower them the temperature of the water coming from economizers and entering the wall tubes.



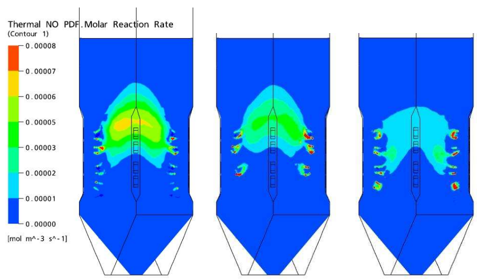
A B C

Figure 4: Temperature distribution inside the boiler (diagonal plane) for cases A, B and C.



A B C

Figure 5: Total heat flux through the walls for cases A. B and C.



A B C

Figure 6: Formation of thermal NOx, cases A, B and C.

Conclusions

The results show that the air at ambient temperature entering in the chamber through the water seal promotes an inversion of the heat flux at the lowest part of the walls, with the pre-heated water coming from the economizer transferring heat to the air. Also the flow dynamics in the boiler is affected by the leakage that creates a distinct upwards flow in the center of the boiler, following the vertical axis. The formation of NOx is enhanced due to this flow that goes directly to the region of higher temperatures prior to react with the pulverized fuel.

References

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.

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, v. 78, n. 4, pp. 2758-2763.

Indrusiak, M. L. S., da Silva, C. V., Beskow, A. B. and Kaehler, J. W. (2007). CFD analysis of the combustion, gas flow and heat exchange processes in a boiler of a thermal power plant. In: Ann. of the ICDERS – 2007, Poitiers.

Kanury, A. M. (1975). Introduction to Combustion Phenomena, Gordon and Beach Science Publishers, New York.

Knudsen, J. G. (1958). Fluid dynamics and heat transfer. Mc Graw Hill.

Kumar, M. and Sahur, 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, 3189-3193.

Li, Z. Q., Wei, Y. and Jin, Y. (2003). Numerical simulation of pulverized coal combustion and NO formation. Chemical Engineering Science, v. 58, pp. 5161-5171.

Silva, C.V., Indrusiak, M.L.S. and Beskow, A.B., (2008). CFD analysis of the combustion, gas flow, and heat exchange processes in a boiler of a thermal power plant. Revista Perspectiva. vol. 32. n118.

Xu, M., Azevedo, J. L. T. and Carvalho, M. G., (2000). Modelling of the combustion process and NOx emission in a utility boiler, Fuel, vol. 79, pp. 1611-1619.

Williams, A., Pourkashanian, M., Jones, J. M. and Skorupska, N., 2000, Combustion and Gasification of Coal, Taylor & Francis, New York.