# CFD SIMULATIONS AS A TOOL FOR FLOW AND THERMAL ANALYSIS IN BOILERS OF POWER PLANTS

**Iván F. Galindo-García[(a)], Ana K. Vázquez-Barragán[(b)], Miguel Rossano-Román[(c)]**

[(a), (b), (c)]Advanced Training Systems and Simulation Department
Institute of Electrical Research
62490, Cuernavaca, México

[(a)]igalindo@iie.org.mx, [(b)]akvb@iie.org.mx, [(c)]rossano@iie.org.mx

## ABSTRACT

Computational Fluid Dynamics (CFD) simulations of the gas flow inside the boiler of a power plant are presented. The CFD simulations can be employed for a very detailed analysis where spatial and local effects can be important. CFD calculations were performed for a 350 MW utility boiler at 100% of total load using either pulverized coal or heavy oil as fuels. Then, two case studies with variation in working conditions are presented: for the first case the effect of the amount of combustion air is investigated and for the second case the failure of one burner is simulated. Simulation of these test cases demonstrate the general capability of the simulator and that CFD methods are recommended as a viable computational tool to evaluate the flow and thermal performance in the gas side of the boiler of a power plant.

Keywords: CFD, boiler modelling, power plant

## 1. INTRODUCTION

A computational model has been developed in order to analyse flow, temperature and species distributions for the gas side of a 350 MW utility boiler. It is assumed that identification of high temperature or high velocity zones will help in the prevention of failures in the boiler walls tubes, superheaters, reheaters and economizers. Failures such as stress rupture, erosion, and thermal fatigue are associated with short- and long-term overheating, fly ash, coal particle impingement, and overfiring or uneven firing of boiler fuel burners. Identifying and correcting the cause of these failures is essential to ensure low availability loss and to eliminate repeat failures. Careful attention has been given to assemble tools that utilities need to eliminate these failures (Dooley and Chang 2000).

One of these tools is numerical modelling, and CFD methods provide a potentially accurate and cost effective tool that can help in the analysis of the gas side of a boiler. Modelling of the chemical and physical phenomena inside the boiler is important for the analysis of failures, because one of the known causes of tube failure is the non-uniform heating of the tubes, which strongly depends on flow and temperature distributions of the combustion gases. CFD is becoming a critical part of the design process of different power plant equipment. During the last 20 years CFD has been applied to study pulverized coal combustion in furnaces to predict the combustion phenomena and to troubleshoot flow, mixing, combustion, and heat transfer problems (Boyd and Kent 1986, Fiveland and Wessel 1988). However, even if in recent years great progress has been achieved, the predictions of CFD models for combustion should be considered as qualitative trends and for parametric analysis (OIT 2002). Nevertheless, careful use of these codes as an engineering tool can help to obtain a reliable prediction of the combustion behaviour of utility boilers.

In the present work computer simulations have been performed to model steady state, 3-D combustion for a 350MW utility boiler burning either pulverized coal or heavy oil. The main aim is to develop a computational tool to investigate flow, temperature and species distributions within the gas side of a boiler that can help to predict zones with abnormal operation for a particular set of operation conditions. The CFD model is part of a simulation tool that integrates the current CFD model with a real time lumped-parameter module of the boiler including all associated systems and controls (feedwater system, steam turbine, controllers, etc.), which permits the user to dynamically simulate different operational conditions and to establish a particular condition to be simulated with the CFD code. In Roldan-Villasana et al. (2010) the real time simulator architecture, the software platform, and mathematical models are fully explained.

This work has been developed at the Advanced Training Systems and Simulation Department (GSACyS, after its name in Spanish) of the Institute of Electrical Research (IIE) in Mexico. The GSACyS has developed several real time simulators for training of power plant operators for the state-owned utility in Mexico (Federal Electricity Commission CFE). For the present work CFE requested the development of a simulation tool that can provide a deeper insight into the physical and chemical phenomena in a power plant boiler in order to be able to identify and predict potential boiler tube failures.

## 2. GENERAL DESCRIPTION OF THE BOILER

A boiler, or steam generator, is a key component in power plants. The boiler extracts energy from the gases product of the combustion, and transfers thermal energy to the water/steam that flows inside multiple sections of heat exchanger tubes (superheater, reheater and economizer). The steam produced in the boiler is then supplied to a steam turbine to generate power.

The boiler under consideration is part of a 350 MW commercial power plant operating in a subcritical steam cycle. The combustion chamber is rectangular in shape (dimensions 12.7 x 14.15 x 45.6 m), and is fired tangentially using five levels of pulverized coal burners or four levels heavy oil fuel burners in each corner. The furnace geometry and burner arrangement are shown in Fig. 1.

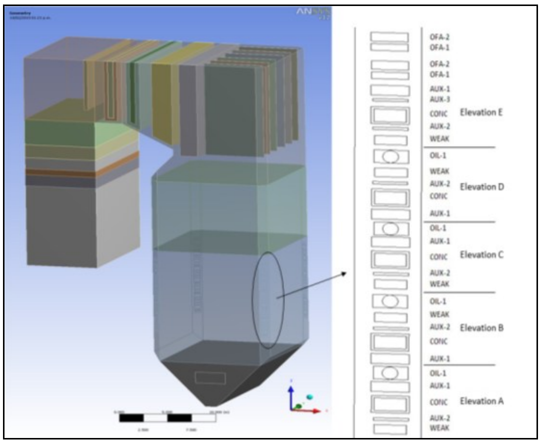


Figure 1: Geometry of the Boiler.

## 3. CFD MODEL

Four stages have been followed for the development of the CFD model: 1. Building the geometry, 2. Mesh generation, 3. CFD simulation and, 4. Post processing and analysis of results. The simulations in the present work were done using the general purpose CFD software "ANSYS FLUENT" (ANSYS 2009a).

### 3.1. Geometry

The solution of any CFD process begins with the generation of the geometry. Technical drawings of the reference boiler have been consulted to generate the computational 3D geometry that represents the actual equipment as closely as possible. The model domain consists of the combustion chamber from the burner nozzles at the furnace corners, up to the exit of the economizer.

### 3.2. Meshing

The accuracy of a CFD simulation depends on the quality of the mesh. Numerical error is a combination of many aspects, for example the grid density, discretization method, and convergence errors (Ferziger and Peric 2002). Numerical error can be minimized using denser grids, higher order discretization methods and suitable time step size. The limitation for these factors is computation time, as time required to get a converged solution for a CFD problem depends directly on the size of the mesh. However in all CFD computations results should be ensured to be grid independent. Generally, it is important to find an optimum between acceptable results and computational time.

In this work a mesh sensitivity analysis was performed in order to evaluate the effect of the mesh on calculations. Beginning with a coarse mesh, simulations have been carried out for different mesh sizes. Target quantities (temperature and velocity) have been obtained as a function of the grid density. The final result of the calculations should be independent of the grid that is used. This is usually done by comparing results of calculations on grids with different grid sizes. Fig. 2 shows the velocity in a horizontal line inside the boiler using different grids sizes.

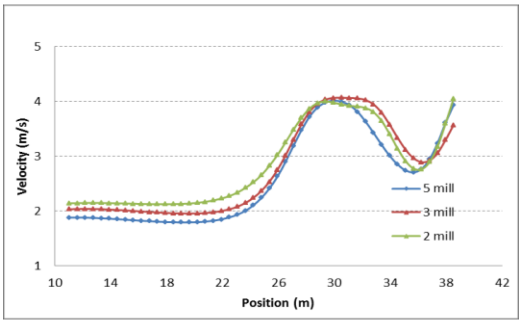


Figure 2: Velocity for Three Different Mesh Sizes.

The computational mesh adopted for these calculations consist of tetrahedral and hexahedral elements and has approximately 3 million elements of unequal size. The regions close to the burners were assigned a denser mesh. Figure 3 shows two views of the mesh.

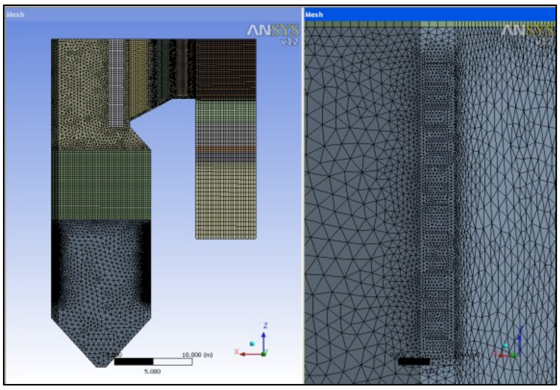

Figure 3: Two Different Views of the Mesh.

### 3.3. CFD Simulation

The CFD simulation involves defining the mathematical models and establishing the boundary conditions for the problem to solve.

The simulation of combustion systems includes modelling a number of complex, simultaneous, interdependent processes such as fluid flow, turbulence, particle transport, combustion and radiation. The time averaged conservation equations (mass, momentum and energy) are solved for predicting the flow, temperature and concentration of gas species. Turbulent quantities are calculated using the standard high-Reynolds-number k-ε turbulence model. Standard wall functions are used to bridge the regions adjacent to solid boundaries, the forms adopted taking $k^{1/2}$ as the velocity scale (ANSYS, 2009b). Lagrangian particle trajectories of the pulverized coal particles or heavy oil droplets are calculated throughout the computational domain. The dispersion of particles due to gas turbulence is predicted using the stochastic tracking model which includes the effect of instantaneous turbulent velocity fluctuations of the gas on the particle trajectories. The P1 radiation model is used to simulate radiation heat transfer. Absorption coefficients of the gas phase are calculated using the weighted-sum-of-grey-gases model (WSGGM). The impact of reacting particles or droplets on the continuous phase can be examined using heat and mass transfer relationships, available in ANSYS FLUENT. For coal particles the model includes particle heating, evolution of volatiles and swelling, char reaction and cooling of the particle. For droplet combustion the droplet evolution includes heating to vaporization temperature, evaporation, boiling and cooling. All models mentioned above have been extensively used for an efficient modelling of the complex phenomena in large-scale boilers. The governing equations for the mean flow in tensor notation are (detailed formulations can be found in the ANSYS FLUENT Theory Guide (ANSYS 2009b)):

Continuity:

$$\frac{\partial(\rho Y_i)}{\partial t} + \frac{\partial(\rho U_j Y_i)}{\partial x_j} = -\frac{\partial(J_i)}{\partial x_j} + R_i + S_i \qquad (1)$$

Momentum:

$$\rho \frac{DU_i}{Dt} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left( (\mu + \mu_t)\left[ \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right] \right) + S_{\mathrm{f}} \qquad (2)$$

Energy:

$$\frac{\partial \rho H}{\partial t} - \frac{\partial P}{\partial t} + \frac{\partial \rho U_j H}{\partial x_j} = \frac{\partial}{\partial x_j}\left( \lambda \frac{\partial T}{\partial x_j} + \frac{\mu_t}{\sigma_t}\frac{\partial H}{\partial x_j} \right) + S_h \qquad (3)$$

The continuity equation predicts the local mass fraction of each species, $Y_i$, in a mixture. Here $R_i$ is the net rate of production of species by chemical reaction and $S_i$ is the rate of creation by addition from the dispersed phase. $J_i$ is the diffusion flux of species $i$. The eddy-dissipation model is used to calculate $R_i$.

A two-step mechanism involving oxidation of volatiles (*vol*) to $CO$ in the first reaction and oxidation of $CO$ to $CO_2$ in the second reaction is employed:

$$vol + v_{O2}O_2 \rightarrow v_{CO}CO + v_{CO2}CO_2 + v_{H2O}H_2O + v_{SO2}SO_2 + v_{N2}N_2$$

$$CO + 0.5O_2 \rightarrow CO_2 \qquad (4)$$

where the stoichiometric coefficients, $v$, are estimated from the ultimate and proximate analyses.

### 3.4. Heat Exchanger Modelling

For the simulation of the tube bundles (superheaters, reheaters, economizers and hanger tubes) located downstream of the furnace, it is not feasible to model each tube individually as this would result in a very large and complicated computational mesh. Instead, a porous media approach is adopted to model pressure drop and heat transfer. The porous media model adds two source terms to the momentum equations, a viscous term and an inertial loss term, which depend on the molecular viscosity and the square of velocity, respectively.

$$S_i = -\left( \frac{\mu}{\alpha} U_i + C_2 \frac{1}{2} \rho |U| U_i \right) \qquad (5)$$

where coefficients $\alpha$ and $C_2$ represent the permeability and the inertial resistance factor, respectively.

For the treatment of the heat transfer inside the porous zone, the energy equation (Eq. 3) is modified in the heat conduction term, using an effective thermal conductivity, $\lambda_{eff}$ that takes into account the fluid, $\lambda_f$ and solid, $\lambda_s$ conductivities and the porosity, $\beta$ of the medium.

$$\lambda_{eff} = \beta \lambda_f + (1-\beta)\lambda_s \qquad (6)$$

The porosity is the volume fraction of fluid within the porous region (i.e., the open volume fraction of the medium). The formula for porosity factor is,

$$\beta = 1 - \frac{\pi D_o^2}{4 S_T S_L} \qquad (7)$$

where $D_o$ is heat exchanger tube diameter, $S_T$ is the transversal length (pitch), and $S_L$ is the axial length. The porous model is employed to model three superheaters (SH1, SH3 and SH4), two reheaters (RH2 and RH3), two economizers (ECO1 and ECO2) and hanger tubes for SH1. Superheater SH2 is modelled as plates and reheater RH1 is not modelled. The geometric data used for the calculation of the porosity of the heat exchangers are given in Table 1.

Table 1: Heat Exchanger Parameters

| | Rows | Tubes | Diam mm | $S_T$, mm | $S_L$, mm | Porosity |
|---|---|---|---|---|---|---|
| SH3 | 26 | 33 | 48.6 | 522 | 58.1 | 0.938 |
| RH2 | 40 | 20 | 63.5 | 348 | 115 | 0.920 |
| RH3 | 80 | 8 | 60.3 | 174 | 115 | 0.857 |
| SH4 | 80 | 8 | 48.6 | 174 | 100 | 0.893 |
| SH1 | 54 | 34 | 54 | 130.5 | 100 | 0.824 |
| ECO 1 | 27 | 12 | 45 | 130.5 | 115 | 0.894 |
| ECO2 | 27 | 12 | 45 | 130.5 | 115 | 0.894 |
| Hanger SH1 | 54 | 5 | 48.6 | 130.5 | 100 | 0.857 |

Total heat absorbed in each exchanger is modelled by adding an energy source term to the energy equation. The value of the source term is calculated based on the percentage of heat absorbed in each heat exchanger. In Table 2, for example, the percentage of heat absorbed in each heat exchanger for 100% load is shown.

Table 2: Heat Absorbed in Each Heat Exchanger

| Heat exchanger | Coal | Heavy Oil |
|---|---|---|
| Boiler Walls (%) | 34 | 36 |
| Superheaters (%) | 33 | 31 |
| Reheaters (%) | 14 | 14 |
| Economizer (%) | 9 | 10 |
| Air Pre-heater (%) (not modelled) | 10 | 9 |
| Total absorbed heat (%) | 100 | 100 |

### 3.5. Boundary Conditions

Once the mesh has been generated, appropriate boundary conditions need to be applied for the surfaces. This step includes defining the inlet, outlet and walls and specifying the zones for the heat exchangers.

Boundary conditions were obtained from the plant's design data sheets. The air and fuel nozzles were the inlets and the boiler final duct after the economizers was the outlet. The boundary conditions required by the model include primary and secondary air flow rates and temperatures, fuel mass flow rates and temperatures, and fuel properties. The outlet boundary was set as a pressure outlet. The boiler walls were assigned wall boundary conditions for flow and thermal properties. Table 3 shows the main boundary conditions for the simulation cases. Properties of fuels, proximate and ultimate analyses, as well as heating value were taken into account to specify the fuels (coal proximate analysis: moisture 9.5%, ash 12.2%, volatiles 31%, fixed carbon 47.3%; ultimate analysis: C 82.5%, H 5.6%, O 8.96%, N 1.8%, S 1.1%, Cl 0.04%; heating value 26,497.27 kJ/kg. Heavy oil: C 83.64%, H 11.3%, S 4.2%, O+N 0.86%; heating value 41,868 kJ/kg).

Table 3: Boundary Conditions

| Parameter | Boundary condition |
|---|---|
| Coal firing | |
| Load | 100% (350 MW) |
| Coal flow rate | 33.786 kg/s |
| Primary air flow rate | 86.666 kg/s |
| Primary air temperature | 70 °C (343 K) |
| Secondary air flow rate | 256.388 kg/s |
| Secondary air temperature | 321 °C (594 K) |
| OFA flow rate | 21.04 % of secondary air |
| Heavy oil firing | |
| Load | 100% (350 MW) |
| Heavy oil flow rate | 21.98 kg/s |
| Heavy oil temperature | 117.4°C (390 K) |
| Air flow rate | 305.55 kg/s |
| Air temperature | 325 °C (598 K) |
| Gas recirculation flow rate | 30.8 kg/s |
| OFA flow rate | 13.78 % of total air |

## 4. MODEL VALIDATION

The data needed for model validation, in particular data for CFD-type calculations, are usually not available in commercial utilities. As stated in Fiveland and Wessel (1988), it is impractical and unlikely that enough experimental data could be collected to provide the detailed information needed for CFD modelling. Therefore the global parameters available from the equipment manufacturer and from routine measurements by plant operators may serve as a guide for model validation. In this context validation refers more to agreement in trends than comparison of absolute values.

For the validation calculations the boiler is assumed to be operating at 100% load. Simulations are compared to some global design parameters available from boiler data, mainly values at the furnace exit such as the average temperature and the average $O_2$ mass fraction. It should be noted that the data is assumed to be an average in a plane at that region. Two different fuels were employed: pulverised coal and heavy oil. The boiler is operated with the lower A–B–C–D levels in operation and the upper E level out of service. For the heavy oil case all four levels are in operation. The tilt angles of the A–D burners were assumed to be 0°. Data from calculations are compared to plant data in Table 4.

Table 4: Comparison Calculations and Reference Plant

| Variable | Calculation | Ref. |
|---|---|---|
| Fuel: Coal | | |
| Flue gas at outlet, kg/s | 372.2 | 376.66 |
| **Gas Temperature** | | |
| Flue gas at furnace exit, °C | 1171 – 1376 | 1007 |
| Reheater outlet, °C | 646 – 826 | 779 |
| Superheater outlet, °C | 466 – 680 | 527 |
| Economizer inlet, °C | 466 – 676 | 524 |
| Economizer outlet, °C | 336 – 576 | 343 |
| $O_2$ at outlet (dry vol %) | 2.6 – 6.5 | 3.6 |
| Fuel: Heavy Oil | | |
| Flue gas at outlet, kg/s | 358.20 | 357.5 |
| **Gas Temperature** | | |
| Flue gas at furnace exit, °C | 1146 – 1606 | 1017 |
| Reheater outlet, °C | 727 – 846 | 773 |
| Superheater outlet, °C | 376 – 562 | 517 |
| Economizer inlet, °C | 376 – 566 | 546 |
| Economizer outlet, °C | 376 – 426 | 352 |
| O2 at outlet (dry vol %) | 6.8 – 8.8 | 1.1 |

It was found that calculated values show a big variation in temperature at each plane, which indicates that temperature is not uniform and that high temperature regions exist within the boiler. This behaviour can be expected due to the very complex flow that develops inside the furnace, as can be seen in Fig. 4, where flow streamlines through the boiler are shown. In general the reference data is within the range found in the calculations, with the exception of the gas temperature at the furnace exit, where the calculated values were significantly higher than plant's data.

However, the general trend in gas temperature as the gas flows through the boiler is similar for calculation and reference data. It is acknowledged, however, that this comparison is only a rough approximation towards model validation and that more plant data is needed for better analysis.

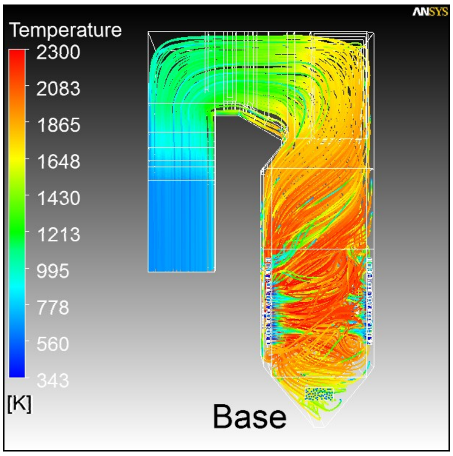


Figure 4: Streamlines Coloured by Temperature

## 5. TEST CASES

CFD simulations can be employed for numerical simulation of existing and possible operation situations and for the purpose of solving problems in power plants operating in working conditions subjected to change (change of the fuel characteristics, load, etc.). In this work two case studies are presented: Case “a”, where the effect of varying the amount of combustion air is investigated, and case “b”, in which the failure of one burner is simulated.

Computations have been conducted in a relatively basic PC (2.66 GHz, 16 GB RAM, 4 cores running in parallel). Computation time has varied approximately from 15, 25 to 40 hours of clock time for meshes of 2, 3.5 and 5 million cells, respectively.

A real time lumped-parameter simulator of the boiler, developed in a parallel work, may be employed to establish operational conditions for the CFD model. The real time module allows the user to perform operational manoeuvres such as increasing and reducing load, or operating air or fuel control valves, with the aim of obtaining the dynamic response of some of the main variables such as pressure and temperature of the steam flow to calculate heat absorbed and temperature of combustion gases. For the cases presented here, however, the conditions were simply assumed and the real time module not used.

### 5.1. Case “a” – Variation of the Air Flow

For the first test case CFD simulations have been performed in order to analyse the effect the amount of air has on the combustion process. It is assumed that variation of the combustion air can have a significant influence on the formation process of some pollutant species such as $NO_x$. Here a small variation of 10% more than and 10% less than the normal amount of air is specified.

Figure 5 shows a plot of the average gas temperature in horizontal cross-sections at different distances in the path of the gas where 0 m is the furnace bottom and 25 m is the furnace exit. The plot shows lower temperature for the case with less air at the bottom and lower temperature for the more air case at the top of the furnace. Figures 6 and 7 show temperature and velocity contours in a horizontal cross-section at the height of burner level D (10 m) for the three cases. Small differences can be observed, for instance in Fig. 6, the high temperature region is more defined as the amount of air increases, which corresponds to the higher velocity, shown in Fig. 7, as more air is being injected.

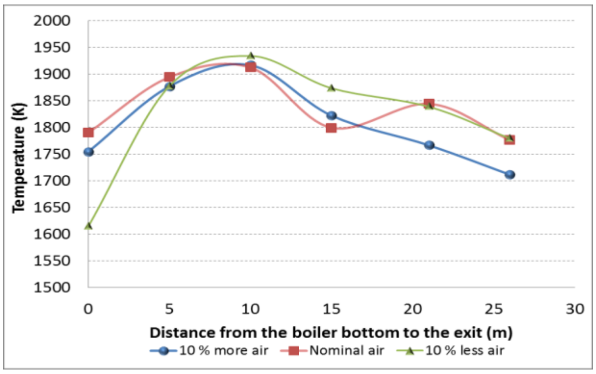


Figure 5: Gas Temperature Varying the Amount of Air

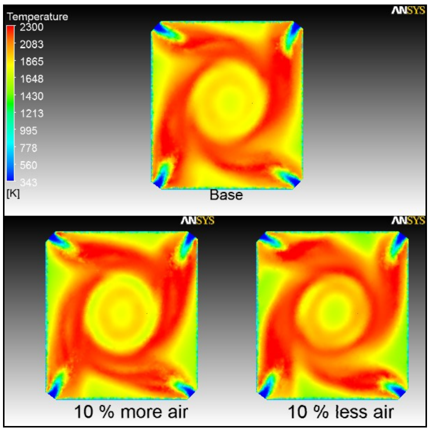


Figure 6: Temperature at 10 m for Test Case “a”

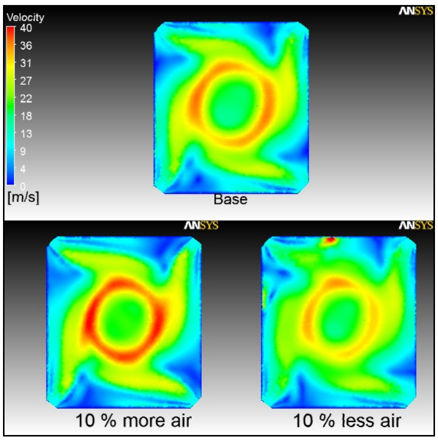


Figure 7: Velocity at 10 m for Test Case “a”

### 5.2 Case "b" – Failure of One Burner

For the second case the boiler is assumed to be at 100% load and a failure on one of the burners is postulated. The failure is specified as injection of air without coal content. Figure 8 shows temperature, velocity and $CO_2$ mass fraction contours at horizontal planes at the failed burner height. It clearly shows the effect on temperature and production of $CO_2$ in the corner with the failed burner. The velocity contour does not show a significant variation because air is still being injected.

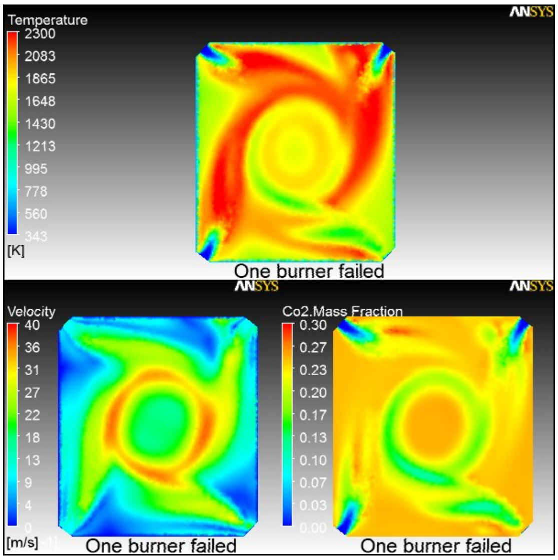

Figure 8: Temperature, Velocity and $CO_2$ at failed burner height (5 m) for Test Case "b"

## 6. CONCLUDING REMARKS

A CFD model has been developed to simulate the gas side of a utility boiler. The CFD model forms part of a simulation tool that includes a real time simulator of the boiler and associated systems that may be employed to establish operational conditions for the CFD model. The CFD model has been validated comparing simulation results to design parameters from the reference plant, where validation refers more to agreement in trends than comparison of absolute values. Two case studies have been presented in which numerical simulations were conducted varying the operational conditions: the amount of air available for combustion, and the failure of one burner. These test cases demonstrate the general capability of the simulator and that CFD methods are recommended as a viable computational tool to evaluate the flow and thermal performance in the gas side of the boiler of a power plant.

## ACKNOWLEDGMENTS

Financial support for this work was provided by CFE (the Mexican utility, Laboratorio de Pruebas a Equipos y Materiales, LAPEM), and IIE (Institute of Electrical Research).

## NOMENCLATURE

| | |
|---|---|
| $C_2$ | inertial resistance factor (1/m) |
| $D_o$ | tube diameter (m) |
| H | specific enthalpy (j/kg) |
| $J_i$ | diffusion flux of species *i* ($kg/m^2s$) |
| k | turbulent kinetic energy ($m^2/s^2$) |
| P | pressure (Pa) |
| $R_i$ | rate of production by chemical reaction (kg/s) |
| $S_i$ | rate of creation from the dispersed phase (kg/s) |
| $S_f$, $S_h$ | source terms ($N/m^3$, $W/m^3$) |
| $S_L$, $S_T$ | axial and transversal length heat exchanger(m) |
| T | temperature (K) |
| $U_i$ | velocity components (m/s) |
| $x_i$ | coordinates direction (m) |
| $Y_i$ | mass fraction of species *i* (-) |
| *Greek letters* | |
| $\alpha$ | permeability ($m^2$) |
| $\beta$ | porosity factor (-) |
| $\varepsilon$ | rate of viscous dissipation ($m^2/s^3$) |
| $\lambda$ | thermal conductivity (j/s m K) |
| $\lambda_{eff}$ | effective thermal conductivity (j/s m K) |
| $\lambda_f$, $\lambda_s$ | thermal conductivity fluid/solid phase (j/s mK) |
| $\mu$ | dynamic viscosity (kg/ms) |
| $\mu_t$ | eddy viscosity (kg/ms) |
| $v$ | stoichiometric coefficients (-) |
| $\rho$ | density ($kg/m^3$) |
| $\sigma_t$ | turbulent Prandtl number (-) |

## REFERENCES

ANSYS, 2009a. *ANSYS FLUENT 12.0 User's Guide*, ANSYS, April 2009.

ANSYS, 2009b. *ANSYS FLUENT 12.0 Theory Guide*, ANSYS, April 2009.

Boyd, R.K., Kent J.H., 1986. Three-dimensional furnace computer modelling. *Proceedings of 21st Symposium (international) on combustion*, Munich, West Germany, 1986. pp. 265–274

Dooley, B. and Chang, P.S., 2000. The current state of boiler tube failures in fossil plants. *Power Plant Chemistry*, 2(4), 197-203.

Ferziger, J.H., and Peric, M., 2002. *Computational methods for fluid dynamics*, 3rd rev ed., Springer, Berlin.

Fiveland, W.A., Wessel RA, 1988. Numerical model for predicting performance of three dimensional pulverize-fuel fired furnaces. *J Eng Gas Turb Power*, 110, 117–126.

OIT, 2002. *Improving industrial burner design with computational fluid dynamics tools: progress, needs and R&D priorities*, Workshop report, U.S. Department of Energy's Office of Industrial Technologies (OIT) and the Sandia National Laboratories (SNL).

Roldán-Villasana, E.J, Cardoso-Goroztieta, Ma.J, Tavira Mondragón J.A, Rossano Román M.B, 2010. Lumped Parameters Modelling of the Waterwalls of a Power Plant Steam Generator, *Fourth UKSim European Symposium on Computer Modelling and Simulation*, pp. 283-288. November 17-19, Pisa (Tuscany, Italy).