AIChE

Comprehensive Process Design Study for Layered-NO_X-Control in a Tangentially Coal Fired Boiler

Wei Zhou, David Moyeda, Quang Nguyen, and Roy Payne
General Electric Energy, 1831 Carnegie Avenue, Santa Ana, CA 92705

DOI 10.1002/aic.12004
Published online September 16, 2009 in Wiley InterScience (www.interscience.wiley.com).

As emissions regulations for coal-fired power plants become stricter worldwide, layering combustion modification and post-combustion NO_X control technologies can be an attractive option for efficient and cost-effective NO_X control in comparison to selective catalytic reduction (SCR) technology. The layered control technology approach designed in this article consists of separate overfire air (SOFA), reburn, and selective noncatalytic reduction (SNCR). The combined system can achieve up to 75% NO_X reduction. The work presented in this article successfully applied this technology to NRG Somerset Unit 6, a 120-MW tangential coal-fired utility boiler, to reduce NO_X emissions to 0.11 lb/MMBtu (130 mg/Nm^3), well under the US EPA SIP Call target of 0.15 lb/MMBtu. The article reviews an integrated design study for the layered system at Somerset and evaluates the performance of different layered-NO_X-control scenarios including standalone SNCR (baseline), separated overfire air (SOFA) with SNCR, and gas reburn with SNCR. Isothermal physical flow modeling and computational fluid dynamics simulation (CFD) were applied to understand the boiler flow patterns, the combustible distributions and the impact of combustion modifications on boiler operation and SNCR performance. The modeling results were compared with field data for model validation and verification. The study demonstrates that a comprehensive process design using advanced engineering tools is beneficial to the success of a layered low NO_X system. © 2009 American Institute of Chemical Engineers *AIChE J,* 56: 825–832, 2010
Keywords: coal combustion, T-fired boiler, SNCR, layered NO_X control, overfire air, reburn

Introduction

As emissions regulations for coal-fired power plants become stricter worldwide, layering combustion modification and postcombustion NO_X control technologies can be an attractive option for efficient and cost-effective NO_X control.[1] Low NO_X burners (LNB), separated overfire air (SOFA), reburn and selective non-catalytic reduction (SNCR) can be combined to provide NO_X reduction comparable to selective catalytic reduction (SCR) performance at a much lower cost. However, the design of a layered-NO_X control system for a large utility boiler requires careful consideration of the furnace flow field characteristics, boiler design, and operational characteristics in the development of the retrofit system specifications. A key objective of the design is to obtain the maximum emissions control performance while maintaining boiler operation and performance at close to baseline levels.

Another critical question in the layered NO_X control application is how SOFA/reburn will impact SNCR system performance. Because the level of NO_X reduction achieved by SNCR is greatly impacted by the CO concentration at the reagent

Correspondence concerning this article should be addressed to W. Zhou at wei.zhou@ge.com

injection locations (Zhou et al., Development and implementation of numerical solution for a SNCR system, submitted), combustion modifications potentially can have a negative impact on SNCR performance by increasing local CO concentrations. To understand this impact and to obtain an optimal design, this study used both a lab scale isothermal physical flow model and a full scale computational fluid dynamics (CFD) model to develop the necessary injector specifications and operating characteristics for the SOFA/reburn system.

Traditionally, isothermal physical flow modeling is used to study the boiler flow characteristics and to optimize SOFA and reburn injection design for achieving good mixing of SOFA and/or reburn fuel with flue gas. The use of physical modeling for simulation of the flow field characteristics of complex combustion systems such as boilers and furnaces has long been recognized.[2] Much of the early literature on physical modeling is concerned with the proper scaling techniques to capture the correct flow characteristics.[3–5] It has also been demonstrated that physical flow modeling could be used to simulate large boiler furnace.[6] The physical flow modeling approach provides a direct visualization of the flow patterns in the boiler. However, this method provides limited insight on combustion characteristics in the furnace. Later works established the use of physical models in conjunction with computational models to simulate large boiler furnaces.[7–9]

In recent years, CFD has become more popular in industrial applications.[7,9,10–14] Proper use of CFD provides not only flow field information, but also combustion and emission characteristics throughout the furnace. CFD simulation also provides a means of evaluating the impact of combustion modifications on postcombustion SNCR performance.[15,16] Accurate CFD predictions require appropriate specifications for model inputs and boundary conditions. The predictions also need to be validated against baseline test data for use with high confidence. The work presented in this article combines both physical flow modeling and CFD modeling approaches for a low NO_X retrofit design. During the study, the results from both models were compared with each other and with baseline field data and performance data for validations and verifications.

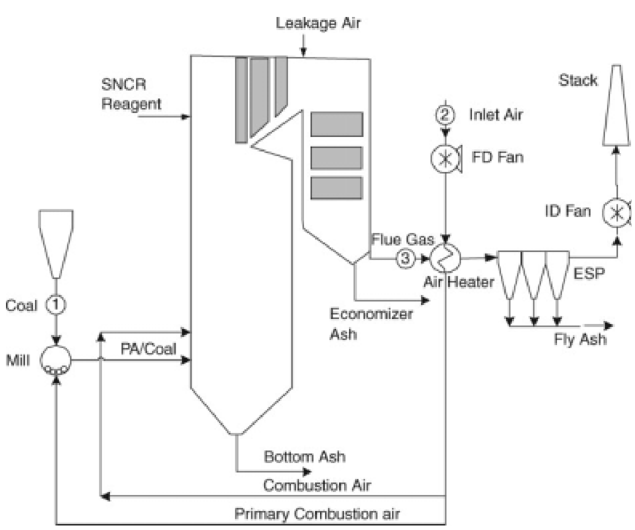

Figure 1. Schematic process flow diagram.

Table 1. Coal Analysis

Parameters	Unit	Value
C	Wt%	72.31
H		5.20
N		1.31
S		0.58
O		6.59
Ash		4.17
Moisture		9.84
Total		100
High heating value	Btu/lb	13,012

The integrated design approach has been applied for over 10 large-scale commercial low NO_X retrofit in the past several years including retrofit of tangentially fired, wall fired, and cyclone fired boilers. This article presents the application of the integrated design approach for design of a layered NO_X control system for NRG Somerset Unit 6. The boiler is a tangential-fired radiant reheat steam generator with a peak generating capacity of 120 MWe gross and fires bituminous coal as the primary fuel. The unit has four mills feeding four burner rows located in each corner of the boiler, for a total of 16 burners. Several years ago, the Somerset plant was equipped with a SNCR system involving multiple levels of urea injection into the convective pass. NRG needed additional NO_X reduction to meet the requirements of the US EPA SIP Call target of 0.15 lb/MMBtu[17] and decided to install a layered control system. The layered NO_X control system consists of SOFA and gas reburn in conjunction with the existing SNCR system. This article summarizes the modeling study of the layered NO_X control system and presents the performance results that validated the design approach.

Boiler Process Conditions

A schematic process flow diagram for the unit is shown in Figure 1. Combustion air is pulled into the system by a forced draft (FD) fan and is preheated by an air heater to 543°K before entering the boiler. Coal is transported and dried by a primary air stream and fed to four elevations of corner burners. The coal analysis is shown in Table 1, and the flow rates for the input and output streams at full load condition are shown in Table 2. The difference between the flue gas flow rate and the sum of coal and airflow rates is the air leakage in the upper furnace, fly ash, and ash to the hopper. Flue gas exits boiler and flows through an electrostatic precipitator (ESP) for particulate removal before flowing to the stack. As shown in the diagram, a SNCR injection system was located in the upper furnace. The objective of the study was to design a SOFA/gas-reburn system that can be placed between the burner pack and the SNCR injection

Table 2. Mass Flow Rates at Full Load Condition

Stream Number	Description	Flow Rate (kg/h)
1	Coal	38,314
2	Total air	404,573
3	Flue gas	451,930

system to further reduce NO_X emissions without reducing the performance of the existing SNCR system.

Integrated Modeling Approach

Isothermal physical flow model

An isothermal flow model of the boiler furnace was built to study the furnace flow field, evaluate the mixing performance of the layered NO_X control system, and validate the CFD model predictions. A subscale (1/14) physical flow model ($7.3 \times 2.3 \times 1.82$ ft^3) was constructed using transparent acrylic to allow smoke and bubble visualization of the furnace flow field, SOFA/reburn jet penetration, and subsequent mixing. The model was designed to be an exact geometric replica of the full-scale furnace, simulating the flow path from the burners in the furnace through the first few banks of the convective pass. A photograph of the model is shown in Figure 2. To ensure similarity of the flow patterns between model and prototype, the burners were scaled using proper techniques,[3–5] and the convective pass sections were scaled to match the full-scale pressure loss coefficients. The SOFA/reburn systems were also evaluated using a Thring-Newby scaling criteria[3] to account for the thermal expansion in the real boiler.

Several visualization and measurement techniques were used to characterize the furnace flow field and evaluate the mixing performance of various SOFA injection configurations. Smoke and bubble tracers were injected through one or more burners to visualize the flow patterns. An 8×10 grid was used for velocity and dispersion measurements. Velocity profiles measured at the SOFA/reburn and nose elevations using hot wire anemometry were obtained to guide SOFA/reburn injector placement and jet penetration. Trace dispersion measurements by a nondispersive infrared sensor (NDIR) were conducted to quantify the flow distribution in the upper furnace. All of these measurements were used in conjunction with CFD results to define the process and hardware design specifications.

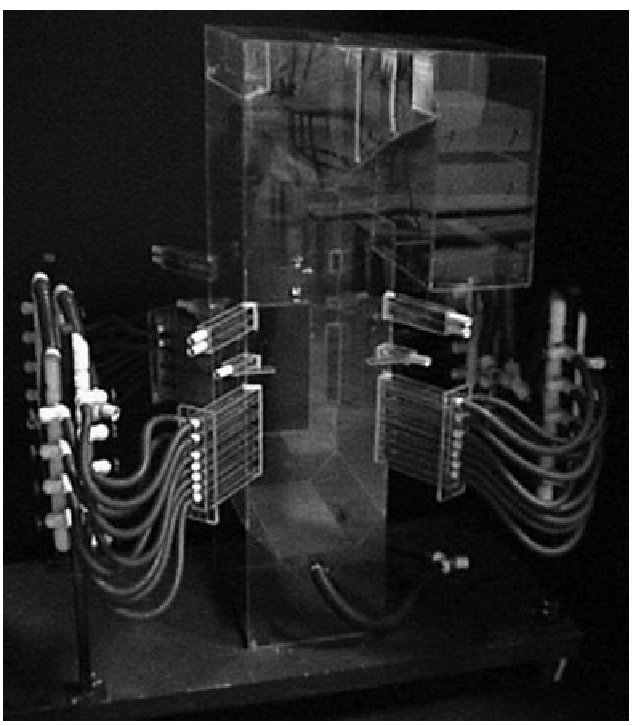

Figure 2. Lab scale furnace model.

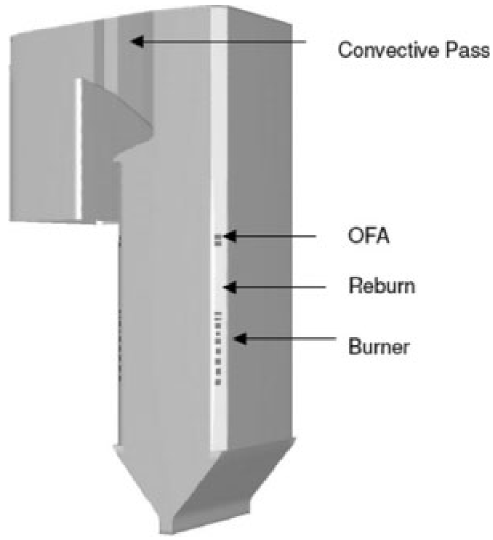

Figure 3. CFD furnace model.

CFD model

A full scale, three-dimensional CFD model was developed for the unit as shown in Figure 3. The modeling domain for the boiler includes the boiler hopper, furnace, burner exits, SOFA and reburn ports, nose, and convective pass, and ends before the economizer. The boiler consists of four levels of coal nozzles, from top to bottom labeled as A, B, C, and D. Combustion air enters the furnace through top and bottom straight air nozzles as well as the offset air ports between the coal nozzles.

The model was discretized to about 600,000 hexahedral cells using a commercial mesh generation tool, ICEMCFD. A finer mesh was used for smaller size nozzles to resolve the details of the flow distribution near the nozzles. A mesh size less than 1 million allows the CFD modeling being an efficient design tool with short turn around time. Transport equations of momentum, energy, and species were solved in each cell using commercial CFD software, FLUENT 6.[18] Physical models such as the k-ε turbulence model, eddy-dissipation combustion model, discrete phase coal devolatilization and char burn out model, discrete ordinate radiation model, and porous media model were used to describe the turbulent coal combustion phenomena and the flow resistance in the boiler. These models have been successfully used in several other coal combustion modeling applications (Zhou et al, Application of numerical simulation and full scale testing for understanding low NO_X burner emissions, submitted). The success lies on the combustion rate calibration and validation against the field data for the baseline conditions. Once the calibration was done, the combustion rates were kept the same for retrofit conditions to predict the performance trend.

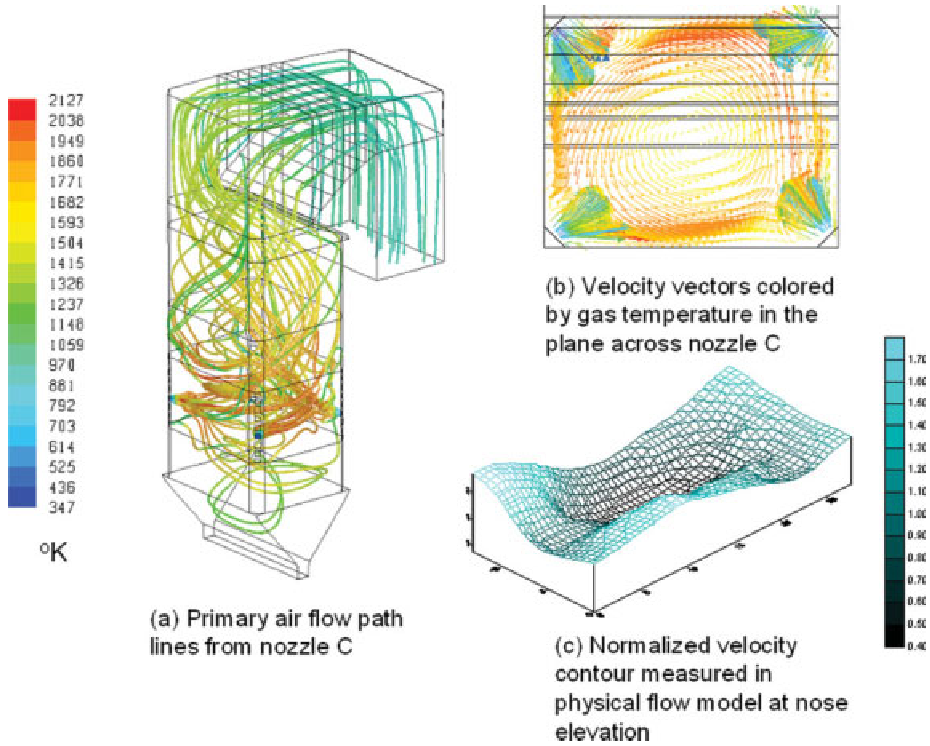

Figure 4. Flow patterns in the boiler.

[Color figure can be viewed in the online issue, which is available at www.interscience.wiley.com.]

The CFD model inputs and boundary conditions, which determine the accuracy of the results, include coal and air flow distributions to the flow inlets, coal fineness, and coal analysis for proper energy balance and mass balance, wall temperatures that take into account of heat exchange between the flue gas and the water wall tubes, heat sinks in the convective pass for proper temperature quench rate, and pressure drops for appropriate flow patterns in the upper furnace. To define the wall temperatures and the heat exchanges in the convective pass correctly, a boiler thermal code[19] was used to calculate the steam side energy balance.

Modeling Results

Baseline study results

A good understanding of the baseline boiler operation is essential to the design of combustion modifications. The baseline model calibration, validation, and verification are necessary steps in the study for building confidence on model predictions. Understanding of the baseline flow patterns also helps to place the SOFA and reburn injectors for achieving good jet penetration and mixing.

Figure 4 shows the flow patterns predicted for the baseline case. A strong counter clockwise vortex is formed in the lower furnace (Figure 4a), which pushes the combusting mixture of coal and air toward the walls, generating high temperature zones near the walls (Figure 4b). The spiral path lines from coal nozzles resemble the flow patterns observed from the physical flow modeling study and are typical of tangentially fired boilers. The flow patterns are characterized by a strong rotational component of the flow with high-velocity near the walls (Figure 4c). The rotational motion persists all the way beyond the nose elevation. The flow then straightens out in the convective pass, with the

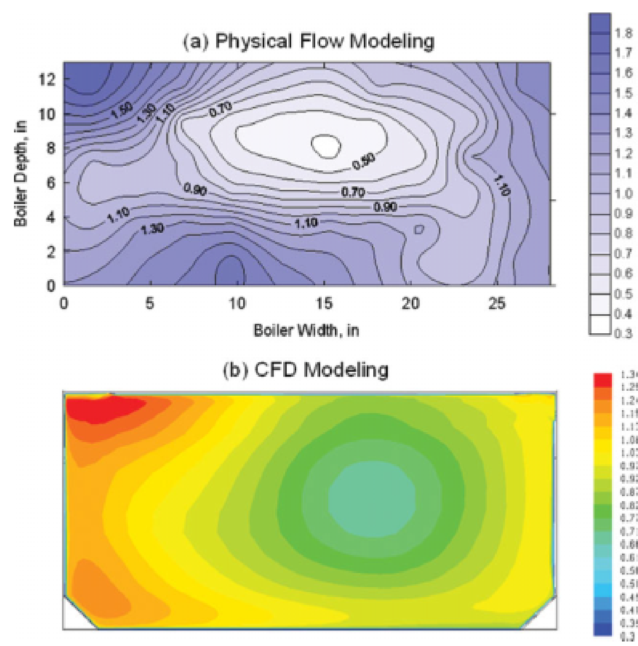

Figure 5. Comparison of velocity profiles at nose plane obtained from physical flow modeling (a) and CFD modeling (b).

[Color figure can be viewed in the online issue, which is available at www.interscience.wiley.com.]

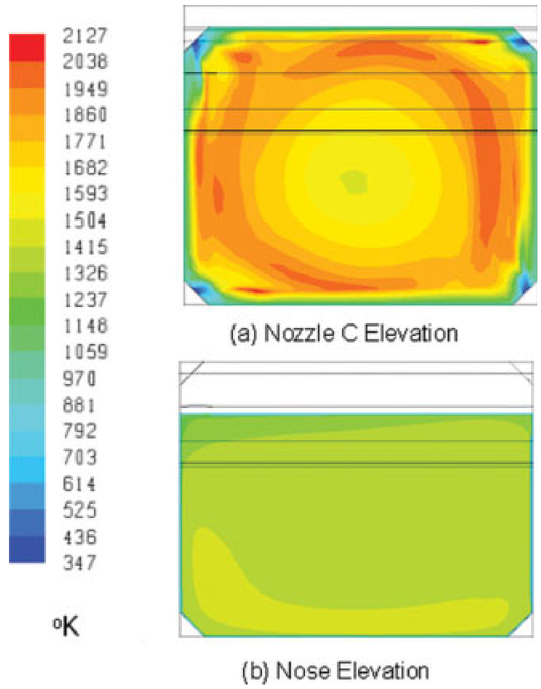

Figure 6. Temperature contours predicted in CFD.

[Color figure can be viewed in the online issue, which is available at www.interscience.wiley.com.]

exchangers modeled as heat absorbing porous media. Figure 5 compares the normalized vertical velocity profile generated from the physical flow modeling with that from the CFD modeling. The two profiles qualitatively agree well with each other.

The temperature profiles predicted by the CFD model are primarily determined by the local fuel heat input, heat release distribution, waterwall temperatures, and flow patterns that distribute the temperature. A plan view of the temperature distribution at the elevation of Coal Nozzle C is shown in Figure 6a. The contour profile shows that the coal ignites at a distance away from the nozzle. The strong vortex pushes the coal flow toward the walls and results in coal combustion near the wall. The gas temperatures in the center of the lower furnace are therefore cooler than those in the wall region.

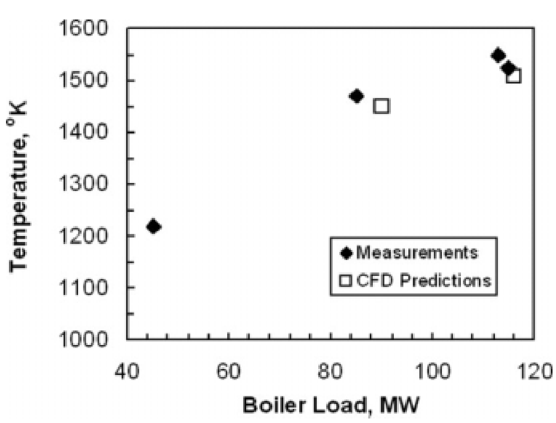

Figure 7. Comparison of measurements with CFD predictions for the furnace exit temperature.

The temperature at the nose elevation, however, is more uniform (Figure 6b) with a spatial temperature variation within 533°K. The predicted average gas temperatures at the nose elevation are about 1500°K for full load and 1450°K for mid load, which are comparable with the baseline High Velocity Thermocouple (HVT) field measurements as shown in Figure 7.

Example profiles for O_2 and CO concentrations are shown in Figure 8. Because the burner zone stoichiometric ratio (SR) is greater than 1 during the baseline operation, excess O_2 is observed throughout the furnace. A top view of the CO distribution at the Coal Nozzle C elevation (Figure 6b) indicates that CO, which is formed in the coal combustion flame zone, is consumed very quickly in the burner zone due to the presence of excess air. To calibrate the combustion rate in the eddy-dissipation model, the CO concentration at the model exit was compared with the stack measurements, assuming that there is no CO reaction downstream of the model exit where the gas temperature is lower than 900°K. The measured CO at stack at full load is around 10 ppm at 2.2% exit O_2. Once the combustion rate is calibrated for the baseline condition, it will be fixed throughout the study to evaluate the retrofit impact on combustion characteristics. The kinetics rate impacts the predicted flame length and local heat flux distribution in the lower furnace. However, the flue gas exit temperature is determined by the total fuel heat input and the total heat loss in the lower furnace and therefore is minimally impacted by the change of the reaction rate.

Boiler retrofit study

The models (CFD and physical flow modeling) were then used to predict boiler operation with 15% total air staging in the SOFA injectors. Figure 9 shows a comparison of the SOFA dispersion profile obtained from the physical flow model and the O_2 concentration profile predicted from CFD at nose elevation. Lower O_2 concentration is observed in the center and higher O_2 concentration is found at the annulus region from both studies. The two dispersion profiles are qualitatively in a good agreement. Both results suggest that the core of the vortex is difficult to penetrate into due to the

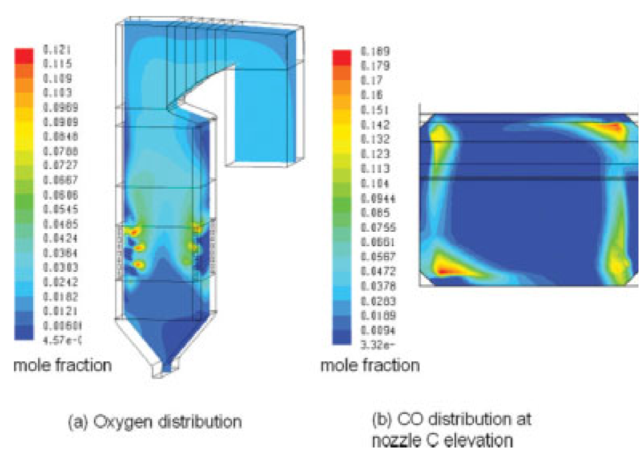

Figure 8. Oxygen and CO contours.

[Color figure can be viewed in the online issue, which is available at www.interscience.wiley.com.]

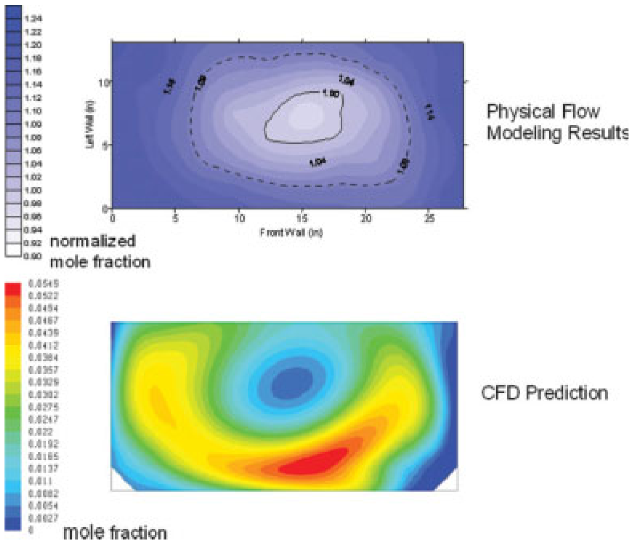

Figure 9. OFA dispersion profile from physical flow modeling and CFD prediction at nose elevation for SOFA only operation.

[Color figure can be viewed in the online issue, which is available at www.interscience.wiley.com.]

large rotating momentum of the bulk flow. The level of SOFA penetration is also limited by the SOFA flow rate and the available windbox pressure. The lower oxygen concentration in the center of the furnace as shown in Figure 8 suggests a higher CO concentration in the same region.

The CFD and physical flow models were also used to predict boiler behavior with 20% reburn. Four gas reburn injectors, placed at each corner, fire at different angles to maximize the reburn gas distribution within flue gas. Figure 10 compares the temperature profiles at the center plane of the furnace for SOFA only and reburn conditions. In the mid furnace, the flue gas temperature is higher with gas reburn operation than with SOFA only operation due to the reburn fuel and CO combustion. As a result, the average flue gas temperature at nose elevation in the reburn case is about 265°K higher than that in the baseline case and about 281°K higher than that in the SOFA only case. A comparison of the average flue gas temperature predicted for baseline, SOFA only and 20% gas reburn operations is shown in Figure 11. The predictions indicate that installation of the SOFA and gas reburn injectors will have a modest impact on gas temperature at and downstream of the nose elevation, where the difference is within 283°K.

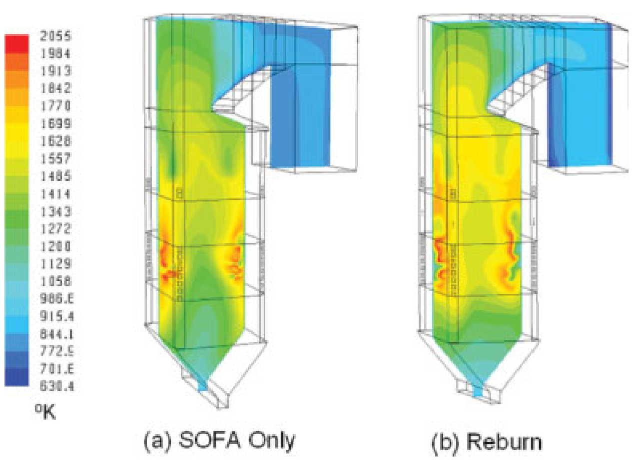

Figure 10. Temperature profiles at the center plane of the furnace.

[Color figure can be viewed in the online issue, which is available at www.interscience.wiley.com.]

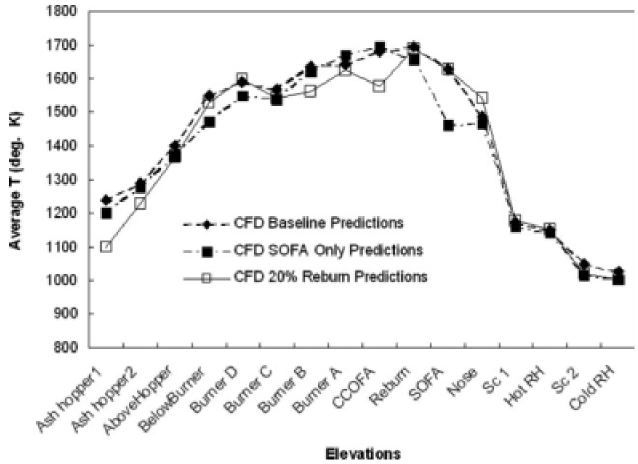

Figure 11. Average gas temperature in the furnace.

Figure 12 depicts the CO mass fraction profiles. Both SOFA only operation and gas reburn operation run at fuel rich conditions in the burner zone below the close-coupled overfire air (CCOFA) ports. However, the SOFA only operation keeps the CCOFA dampers closed, whereas the gas reburn operation keeps the CCOFA dampers fully open. Therefore, the CO distribution profiles immediately above the burner pack have different characteristics for the two operating conditions. High CO concentrations are found near the wall above the CCOFA ports for SOFA only operation. A high CO region is only found at the furnace center in the gas reburn case. Figure 13 shows the average CO concentrations along the furnace. SOFA only operation has the highest CO concentration in the furnace among the three operating modes. However, both SOFA and reburn operations were

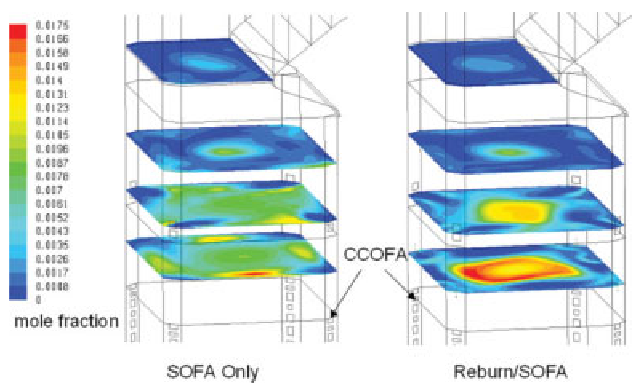

Figure 12. CO profiles at different elevations in 20% reburn study (mole fraction).

[Color figure can be viewed in the online issue, which is available at www.interscience.wiley.com.]

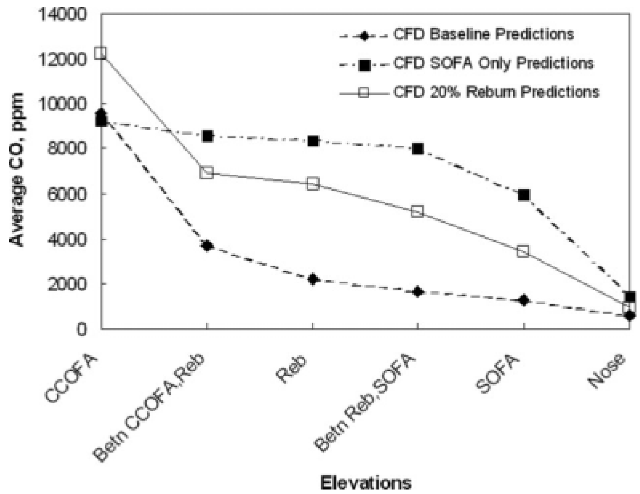

Figure 13. Average CO concentration in the furnace.

predicted to have a good control of CO emissions close to the level of the baseline emissions.

Impact of combustion modifications on SNCR performance

The existing urea-based SNCR system at Somerset injected the urea solution at multiple levels into the boiler. The overall chemical reaction for reducing NO_X with urea is:

$$NH_2CONH_2 + 2NO + 1/2O_2 => 2N_2 + CO_2 + 2H_2O$$

Urea de-NO_X chemistry is greatly impacted by the local flue gas temperature and CO concentration. Figure 14 shows the results of pilot scale SNCR testing with urea and plots the ratio of outlet to inlet NO_X emissions for tests where the reagent injection temperature is varied. This figure illustrates the sensitivity of the SNCR process to flue gas temperature and the nominal temperature window for optimal performance. The SNCR performance is also sensitive to local CO concentrations (Zhou et al., Development and implementation of numerical solution for a SNCR system, submitted).

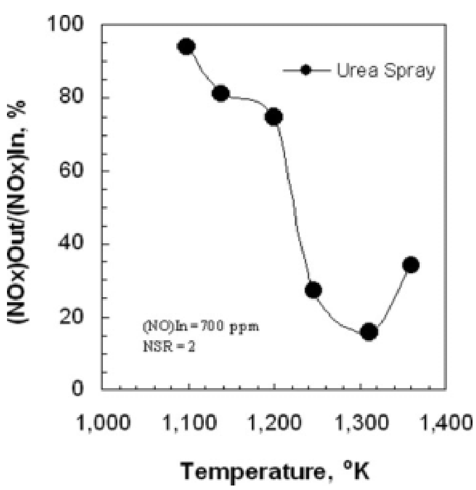

Figure 14. Impact of gas temperature on urea-based SNCR process performance.

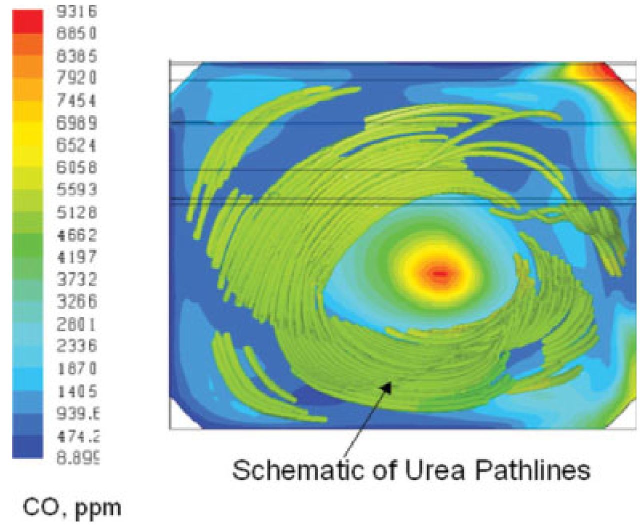

Figure 15. Urea pathlines vs. CO concentration distributions.

[Color figure can be viewed in the online issue, which is available at www.interscience.wiley.com.]

Local CO concentrations of 500 ppm or higher can impose a negative impact on the SNCR performance.

The process design study performed in this article verified that the combustion modifications would have modest impact on flue gas temperature prior to the urea injection locations and that the CO concentration remains low at the outside of the core of the flue gas vortex as shown schematically in Figure 15. Therefore, the combustion modifications were not expected to impact the performance of the existing SNCR system.

Design performance

The layered system was installed and evaluated. The results of several controlled tests are presented in Figure 16, which plots NO_X emissions as a function of stoichiometry in the reburning zone. An overall NO_X reduction of 75% from baseline was achieved, in which an incremental of 40% is due to SNCR. As projected in the process design study, SNCR could be applied in conjunction with gas reburning with minimal impacts on SNCR performance.

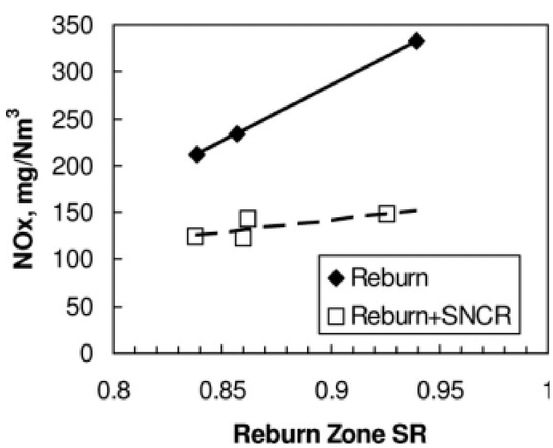

Figure 16. System performance.

The test data also showed that the CO emissions were lower than 90 ppm during both SOFA/SNCR and reburn/SNCR system performance tests.

Conclusions

The study successfully integrated CFD and physical flow modeling to design a layered NO_X control system. The modeling tools were used to predict furnace temperature, oxygen, and CO concentration distributions at baseline, SOFA only and reburn operation conditions. The study optimized the reburn and OFA design to minimize the impact on boiler and SNCR system performance. The performance tests show that as predicted in the study, the combustion modification system did not reduce the SNCR system performance. A total of 75% of NO_X was achieved.

As mentioned earlier that the integrated design approach has been applied to over 10 commercial low NO_X retrofits. This approach requires a higher process design cost than using CFD modeling or physical flow modeling alone. However, the integrated design approach provides higher confidence on the design quality and also lends validation data to transition the integrated design approach to CFD only design approach in future process design with high confidence.

Literature Cited

1. Payne R, Gauld D, Hartsock D, Stonkus P, Folsom B. *Layered Technology Approach Reduces NO_X to Less Than 0.15 lb/MMBtu. Proceedings of Power-Gen International*, November 30-December 2, 2004, Orlando, FL.
2. Beer JM. The significance of modeling. *J Inst Fuel*. 1966;39:466–473.
3. Thring MW, Newby MP. *Combustion Length of Enclosed Turbulent Jet Flames, Fourth Symposium (International) on Combustion*. Baltimore, MD: Williams and Wilkins, 1953.
4. Robertson AD. Scaling and modeling criteria for burner design with swirl addition. *J Inst Fuel*. 1996;39:335–342.
5. Davison FJ. Nozzle scaling in isothermal furnace models. *J Inst Fuel*. 1968;41:470–475.
6. Anderson DK, Bianca JD, McGowen JG. Recent developments in physical flow modeling of utility scale furnace. Industrial Combustion Technologies, American Society for Metals, pp. 117–126, 1986.
7. Honnery DR, Kent JH. Furnace flow modeling: physical and computational. *J Inst Energy*. 1989;169–177.
8. Wirtz S, Kremer H. Numerical prediction of boiler furnaces supported by down-scaled physical laboratory models. In: *Proceedings of the IFRF/GRI/AFRC 7th Topic Oriented Technical Meeting on Scaling and Mathematical Modeling of Combustion Systems*, Chicago, IL, 1993, pp. 27–29.
9. Hules KR, Marshall JJ, Thamaraichelvan P. Computer and physical flow modeling in corner fired furnaces. In: *Proceedings of the IFRF/GRI/AFRC 7th Topic Oriented Technical Meeting on Scaling and Mathematical Modelling of Combustion Systems*, Chicago, IL, 1993, pp. 27–29.
10. Wirtz S, Kremer H. Numerical predictions of boiler furnace supported by down-scaled physical laboratory models. In: *Proceedings of the IFRF/GRI/AFRC 7th Topic Oriented Technical Meeting on Scaling and Mathematical Modeling of Combustion Systems*, Chicago, IL, 1993, pp. 27–29.
11. Zarnescu V, Pisupati SV. An integrative approach for combustor design using CFD methods. *Energy Fuels*. 2002;16:622–633.
12. Toit JD, Kruger JH. Integrated systems CFD analysis applied to boiler simulation. In: *Annals of the Assembly for International Heat Transfer Conference*, 2006.
13. Breen BP, Urich JA. Computational fluid dynamics (CFD) modeling of boiler furnaces for engineering analysis. PEPSE Users Conference, June 1994 http://famos.scientech.us/Papers/1994/1994section17.pdf.
14. Cremer M, Valentine J, Shim H, Davis K, Adams B, Letcavits J, Vierstra S. CFD-based development, design, and installation of cost-effective NO_X control strategies for coal-fired boilers, The Mega Symposium: EPRI-DOE-EPA Combined Utility Air Pollutant Control Symposium, Washington, DC, May 2003.
15. Cremer MA, Heap MP, Ciarlante V, Zoccola MA. CFD modeling of SNCR performance in Conectiv's Indian River Units 3 and 4. Presented at the 1999 U.S. DOE Conference on SCR and SNCR for NO_X control. http://www.netl.doe.gov/publications/proceedings/99/99scr-sncr/cremerl.pdf
16. Cremer MA, Adams BR, Bhamidipati VN, O'Connor DO, Broderick G. Design, installation, and testing of rich reagent injection in Conectiv's B.L England Unit 1. Presented at the 2001 U.S. DOE Conference on SCR and SNCR for NO_X control. http:// www.netl.doe.gov/publications/proceedings/03/scr-sncr/Final_Cremer.pdf
17. Culligan K, Krolewski MJ. EPA update on NO_X regulation. 2002 Conference on Selective Catalytic Reduction and Non-Catalytic Reduction for NO_X Controls, Pittsburgh, PA, Maya 15-16, 2002. http://www.netl.doe.gov/publications/proceedings/02/scr-sncr/krolewskisummary.pdf
18. FLUENT User's Guide, Ansys Inc., Lebanon, NH. http://www.ansys.com.
19. Wu KT, Li B, Moyeda DK, Payne R. A PC-based computer model package for boiler reburning/co-firing. Presented at the ASME International Joint Power Generation Conference & Exposition, Kansas City, MO, October 17-20, 1993.

Manuscript received Apr. 16, 2009, and revision received Jun. 12, 2009.