Fourth International Conference on CFD in the Oil and Gas, Metallurgical & Process Industries
SINTEF / NTNU Trondheim, Norway
6-8 June 2005

APPLICATIONS OF CFD MODELING IN CANADIAN INDUSTRIES

Eddy H. CHUI

CANMET Energy Technology Centre, Natural Resources Canada
1 Haanel Drive, Ottawa, Ontario, Canada K1A 1M1

ABSTRACT

Applying CFD to targeted Canadian industries has been a learning experience. The focus of the paper is to describe this experience with actual full-scale examples in the power generation, steel making and refining sectors, highlighting specific important elements in the application approach that have resulted in a growing general acceptance in utilizing CFD to generate information to resolve real operational issues. The three elements are: 1) credible model input, 2) validation and customisation and 3) real payback due to enhanced understanding. The intent is to share practices that may promote the acceptance and utilization of CFD in practical combustion engineering problems.

Key words: CFD, Modeling, Canadian Industries

INTRODUCTION

For about two decades, CANMET Energy Technology Centre (CETC), the research branch within the Energy Department (Natural Resources Canada) of the Federal Government of Canada, has been actively developing a CFD modeling capability to assist Canadian industries on diagnostic analyses, performance enhancement studies of new energy cycles, alternate operating conditions, and innovative equipment designs of combustion systems.

In the mid-1980s, CFD already emerged as an attractive tool to tackle practical full-scale engineering problems. Robinson (1985), Abbas (1986) and Boyd (1986) are early examples of CFD applications in the power generation sector. However, when CETC attempted to introduce CFD to Canadian industries in the early 1990s, most industries related to the energy sector did not readily embrace it despite its potential to provide useful technical information for optimisation, feasibility analysis and risk assessment purposes. CETC explored the gap between CFD research and industry practice in a specific problem in Runstedtler (2003). The objective of this paper is to present more fully our experience in applying CFD to enhance the operations of Canadian industries. The application approach, developed in a trial-and-error manner, is a response to the scepticism expressed by the industries towards CFD. Presently, this approach has shown to be effective in assisting industries to exploit the power of CFD but is still being refined and adjusted in new applications. The paper will highlight three elements in the application approach that have contributed to the growing acceptance of CFD by some targeted Canadian industries. The focus is on how CETC met the requirements of these elements using three different examples. To respect the wishes of the industries, model theories and results that are proprietary in nature will not be presented in full. The purpose of the paper is to share practices that enhance acceptance of CFD and not to present a modeling methodology for a specific application.

ELEMENT #1 - CREDIBLE MODEL INPUT

500MW Gas-Fired Utility Boiler

A 500+MW$_e$ utility boiler was retrofitted to increase fuel flexibility. Natural gas firing was added to its original oil-firing capability to take advantage of fluctuating fuel costs and increase the potential to reduce CO$_2$ emissions by switching to a lower carbon fuel. The unit could be fully natural gas-fired or oil-fired or a combustion of both from quarter load to full load. However, the introduction of natural gas firing to a unit not originally designed for such conditions generated new operational issues. Some were resolved by field trials but others would require a more in-depth understanding of the combustion process in order to determine the appropriate solutions. The goal of the CFD study was to model various combustion arrangements to reduce furnace outlet temperature and NO$_x$.

Lack of Input Conditions and Proposed Remedy

In this tangentially fired utility boiler, the natural gas combustion involves two stages of mixing between air and natural gas: first in the fuel air compartments before exiting the burners and subsequently in the boiler with air from the auxiliary air compartments. It is expected that the natural gas, undergoing a large drop in pressure, enters the fuel air compartment at a high speed and mixes with air without burning prior to entering the combustion chamber to avoid overheating the burner face. To properly model this partially premixed natural gas combustion in the boiler, the airflow rates at the various compartments (Figure 1&2) should be input as inlet boundary conditions to the model. However, these individual airflow rates were not known at the outset of the CFD study because only the total amount of air going into the windbox is controlled and recorded at the plant. CETC presented this dilemma to the power company and suggested a numerical approach to first determine the distribution of air over the various compartments prior to performing any boiler simulation.

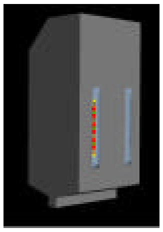

Figure 1: Schematic of the tangentially-fired boiler (with one burner assembly highlighted in multi-colours.

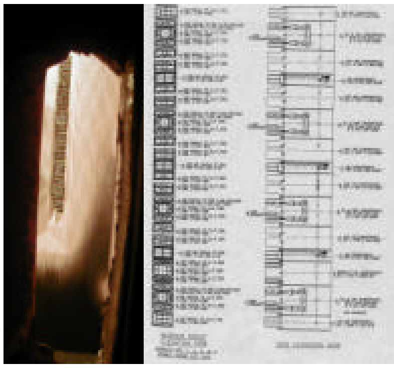

Figure 2: Burner assembly at the corner of a tangentially fired boiler.

Figure 2 shows the front and side views of the burner assembly at each of the four corners of the tangentially fired unit. The pressure drop characteristics over each type of compartment are estimated by a CFD analysis taking into account the actual geometry and damper position.

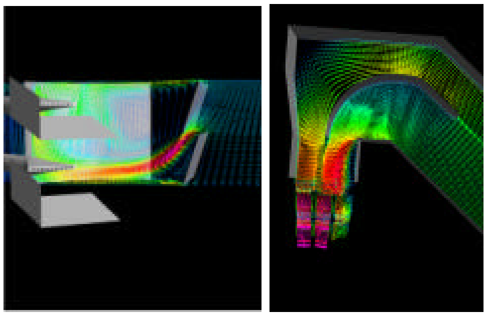

Figure 3: Velocity vectors at the fuel air (left) and auxiliary air (right) compartments.

Figure 3 shows the sample results of the air velocity vectors within the fuel air compartment (going over a set of damper) and auxiliary air compartments (adjusting to a turning vane). By repeating the CFD analysis over a range of flow rates, the relationship between air pressure drop and flow rate is characterized for each compartment type. The correct airflow through each compartment is obtained from the pressure drop versus flow rate curves at the common pressure drop that yields individual flow rates summing to the recorded amount of air for the windbox. Together with these flow rates, the shape of the air velocity profile at each nozzle outlet (Figure 3) were incorporated as input to the boiler simulation. (Refer to Chui (2004) for more details of this CFD study.) The simulation of the base case was completed without any prior knowledge of the measured data at furnace outlet.

Importance to CFD Acceptance

The end user of the CFD results (a power company in this case) was allowed to appreciate the importance of model input conditions and totally involved in deciding the approach to determine the missing input information. Also, the end user was assured that the simulation could not have been "tuned" to produce the expected results (purposely withheld from the modellers). The fact that the calculated outlet NO_x level was within 3% of the measured value not only provided a confidence limit for the model but also demonstrated that the model results were credible and could be used for industrial purposes. An optimal improvement strategy was identified by the CFD study.

ELEMENT #2 - VALIDATION & CUSTOMIZATION

Pulverized Coal Injection (PCI) into Tuyeres

Pulverized coal injection into blast furnace tuyeres is an innovative technology that reduces coke consumption and enhances blast furnace operation in steel making (Figure 4). A tuyere is a water-cooled nozzle (about 0.4m long, 0.2m ID) that connects the blowpipe to the blast furnace. A typical blast furnace has about 24 blowpipe-tuyeres. Since coke is made by pyrolysing coals in large ovens, reducing coke consumption through PCI also results in the decrease of greenhouse gas carbon dioxide and sulphur dioxide.

A Canadian steel manufacturer has been successfully implementing the PCI technology. However, when certain types of coal are used, unburnt coal char is found to escape through the blast furnace, limiting the injection rate. To ensure a wide range of inexpensive coals be usable in PCI, natural gas is introduced to enhance the combustion conditions. The goal of the CFD study is to develop optimal approaches for co-injecting pulverized coals and natural gas into blast furnace tuyeres to minimize coke consumption, emissions and cost.

Uncertainty in Customized Model Performance

The tuyere is a challenging combustion environment to simulate. Blast air is typically pre-heated up to 1200^{o}C and travels at high speed through the blowpipe and tuyere prior to entering the raceway of the blast furnace. As pulverized coal and natural gas are injected into the blast air, they have a very short residence time in the tuyere. The proposed modeling approach (Chui (2003), different from the models for pure PCI applications like Takeda

(1997) and Picard (2001), is customized to account for this fast-moving, two-phase, multiple-fuel combusting flow process. However, the model performance is unknown. It is risky to directly apply it to simulate an industrial scale blast furnace without validation.

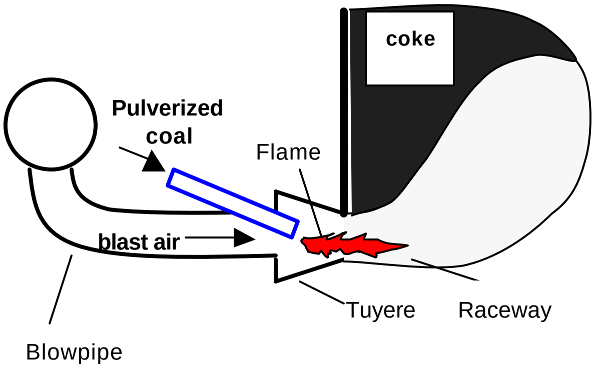

Figure 4. Blast furnace combustion zone with pulverized coal injection

Pilot-Scale Validation

CETC has developed a pilot plant facility that simulates blast furnace blowpipe-tuyere conditions (Figure 5). Its main component is a cylindrical reactor (1m in length, 0.03m ID) consisting of a heavily insulated, refractory-lined inner core and a steel shell. Figure 6 shows the natural gas and coal injection ports positioned at 45° to the axis of the reactor and the sampling ports near the end of the reactor. Inside one of the two sampling ports is a thermocouple located at the edge of the reactor wall to continuously monitor the presence of reaction, while the other port is used for either solid or gas samplings.

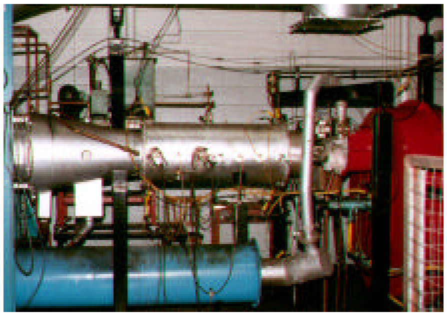

Figure 5: CETC pilot plant reactor for co-injection of coal and natural gas.

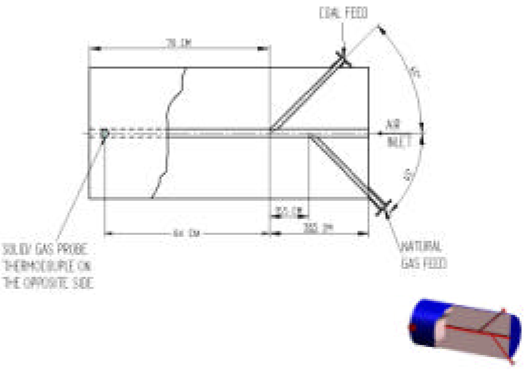

Figure 6: Schematic of reactor for co-injection of coal and natural gas with sampling ports.

As shown in Figure 6, enriched air (25% O_2), preheated to 740°C, was introduced into the reactor. At roughly 0.23m from the air inlet, natural gas was injected into the air stream through a small nozzle (0.0015m ID) to enhance penetration. Pulverized coal together with a small amount of carrier gas were introduced 0.15m downstream of the natural gas injection through a 0.004m ID coal lance. Under appropriate operating conditions, the system could self ignite and sustain combustion inside the reactor. At the sampling station close to the reactor exit, gas concentrations of CO, O_2 and CO_2 were collected at three locations across the traverse of the reactor diameter. Solid samplings were also obtained at the same locations to determine the coal weight loss (due to volatile matter release or burnout) after passing through the reactor. This particular coal-natural gas injection system does not exactly emulate the industrial unit but represents an interesting injection order with data to validate the theoretical modeling approach customized for this problem.

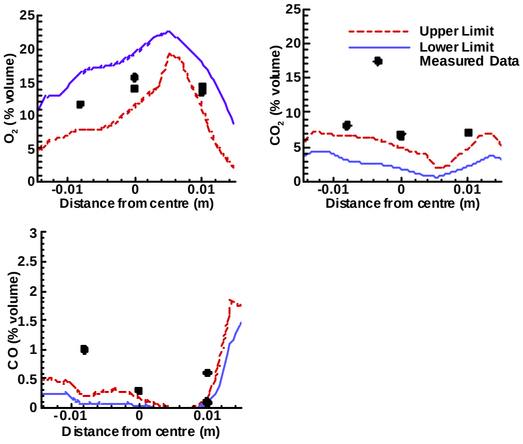

Figure 7: Comparisons between predicted and measured species concentrations along sample line.

Figure 7 shows the comparisons between predicted and measured O_2, CO_2 and CO levels along the sample line close to the exit of the reactor. Because of the oscillating nature of the flame inside the reactor, the predicted results (corresponding to the upper and lower size limits of the combustion zone) are plotted to compare with the time-averaged experimental data. For all the species, the predicted variation along the sample line matches well with the measured one. The O_2 measurements fall well within the range of the predicted values but the CO_2 and CO measurements are both slightly higher than the computed results. Given that the species concentrations were observed to vary quite widely during sampling, the agreement between predicted and measured values is quite reasonable.

Importance to CFD Acceptance

The pilot-scale validation experiments provided some concrete data to gauge the performance of the customized model for a challenging problem. This was important to the steel company because it would have been very difficult to collect such data in the industrial unit. Also, the company appreciated that they were made aware of the limits of the base commercial CFD code; an effort was made to improve the theoretical basis of the simulation but the uncertainty of the model performance was not hidden. Currently the model is being used to investigate the designs of various injection schemes and has been shown to be much more cost-effective than building and trying different prototypes in the actual unit. Field trials can be kept to a minimum and are implemented only for the most promising designs.

ELEMENT #3 - REAL PAYBACK DUE TO ENHANCED UNDERSTANDING

Refinery Furnace for Heating Process Fluid

The non-uniform heat distribution in the radiant section of a nine-burner heater (Figure 8) was a concern to a Canadian refinery because the two streams of process fluid going through the unit received uneven heat transfer and reached the next process at unacceptably different temperatures, compromising production and safety. Repeated field efforts to balance the fuel input to the furnace did not seem to fully correct the problem. The main goal of the CFD study was to investigate the cause of non-uniform heat distribution.

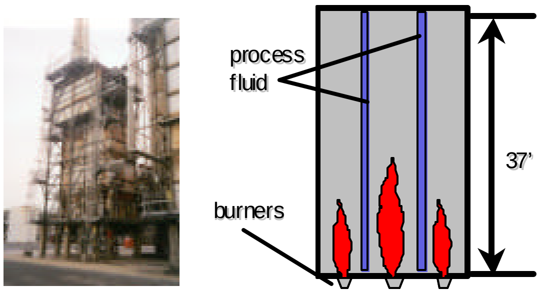

Figure 8: Picture of refinery furnace and the schematic of its cross-section.

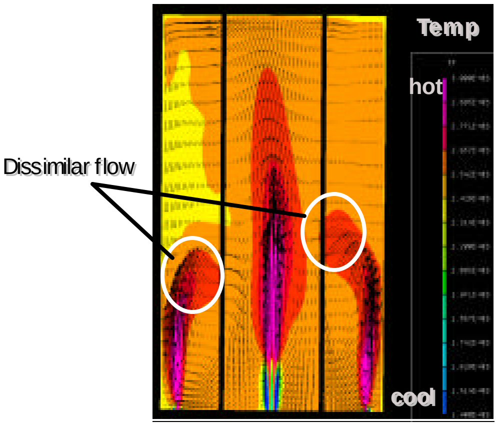

Figure 9: Furnace cross-section colored by gas temperature, with flow pattern superimposed.

Cause of Uneven Heat Distribution

Even with identical fuel and airflow rates to each of the six outer burners on the two sides of the furnace, the simulation shows that the temperature distribution in the upper part of the furnace is far from symmetric (Figure 9), just as observed in the field. This asymmetric temperature distribution is the result of the unstable flow characteristics inside the furnace. By design, the jet flames from the three burners in the centre of the furnace posses a higher momentum than those from the six outer burners. If the conditions on the two sides of these high momentum jets are even slightly different, they will hit the furnace ceiling off-centre and generate an unbalanced flow pattern and hence, an uneven heat distribution to the two streams of process fluid. Perfect symmetry can be achieved only if the fuel and airflow rates on the outer burners are balanced and the flame holders have identical shape and orientation. It is unrealistic to expect these conditions to be fulfilled in real practice.

Importance to CFD Acceptance

The plant engineers were able to exploit the new understanding provided by CFD to make modifications on the burners. The temperature differential between the two process streams dropped from hundreds to tens of °F. Energy consumption of the furnace was reduced due to improved efficiency. More importantly, the unit is now able to handle an elevated throughput, translating into a revenue increase of almost CDN$300,000 per year.

SUMMARY REMARKS

Three selected elements in the CFD application approach practised by CETC are illustrated by three industrial examples. In principle, these elements are fairly obvious and self-explanatory. However, when they are actually implemented in real practical situations, we have witnessed a significant positive impact to the general acceptance of CFD by some Canadian industries, in sharp contrast to the scepticism and lack of interest towards CFD when it was first introduced to the same industries. In hindsight, this should not have been surprising because engineers in many combustion related sectors have always been relying heavily on experience to operate units with large-scale combustion to ensure production targets and safety. CFD for combusting flows is often viewed as an academic tool for research purposes. Our experience shows that with a cautious and open application approach, it is possible to exploit CFD in full-scale combustion engineering problems, despite the imperfections in various theoretical aspects of combustion CFD. In CETC, our current efforts are directed towards improving the theoretical basis of specific selected areas and developing new CFD tools that are better suited to the response time requirements and the budget and payback constraints in the industrial sector.

ACKNOWLEDGEMENTS

This work is supported by the Panel of Energy Research and Development (PERD) Program of the Federal Government of Canada. All the CFD applications described in this work were completed with CFX-TASCflow, currently owned by ANSYS. The author appreciates the long-term collaborations with the developers of CFX-TASCflow to make customisations of the CFD code for specific applications possible. Thanks are extended to colleague A. Runstedtler at CETC for his assistance in this study.

REFERENCES

Abbas, A. S., Lockwood, F.C. (1986). "Prediction of Power Station Combustors". *Twenty-first Symp. (Int.) on Combustion*, Munich, The Combustion Institute, August.

Boyd, R. K., Kent, J.H. (1986). "Three-dimensional Furnace Computer Modelling". *Twenty-first Symp. (Int.) on Combustion*, Munich, The Combustion Institute, August.

Chui, E. H., Runstedtler, A., Majeski, A., Leach I., Macfadyen, N. (2004). "CFD Modeling of a 500MW Gas-Fired Utility Boiler". *2004 International ANSYS Conference*, Pittsburgh, May 2004.

Chui, E. H., Scott, K.A., Harrison, F.W., Macfadyen, N.K. (2003). "Modeling the Co-Injection of Coal and Natural Gas in a High Speed Environment". *Seventh International Conference on Energy for a Clean Environment*, Lisbon, July.

Picard, M., Usinor, I., Romaine, V. (2001). "Pulverized Coal Combustion in the Blast Furnace Raceway using a 3D Numerical Simulation". *2001 Ironmaking Conference*, Baltimore, March.

Robinson, G. F. (1985). "A Three-dimensional Analytical Model of a Large Tangentially-fired Furnace." *J. of the Inst. of Energy*, **LVIII**(436): 116-150.

Runstedtler, A., Chui, E.H., Majeski, A.J., Trottier, S., Brunet, D. (2003). "Modeling a Refinery Furnace". *American Flame Research Committee International Symposium*, Livermore, October.

Takeda, K., Lockwood, F.C. (1997). "Integrated Mathematical Model of Pulverized Coal Combustion in a Blast Furnace." *ISIJ International*, **37**(5): 432-440.