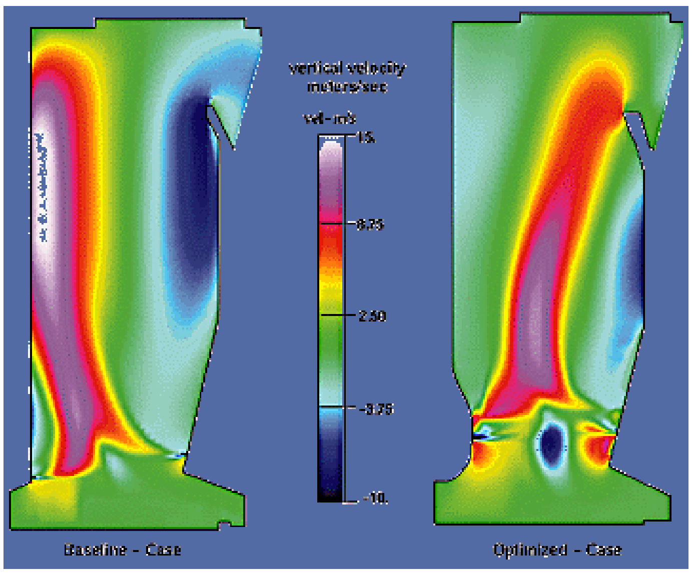
Combustor Design and Modification Integrates Computational Fluid Dynamics

By Paul Chapman, Consulting Engineer, ABB Power Plant Laboratories
ABB Combustion Engineering, Inc. , Windsor, Connecticut



Vertical Velocity Distribution. The baseline condition illustrates the flow pattern expressing the vertical velocity components over the range front-10 to 15 m/s. The baseline case with only a single elevation of overfire air produces a high velocity flow channel attached to the front wall with an associated recirculation down the rear wall of the main combustor section. The optimized case includes a revised overfire air configured to centralize the vertical flow region. The peak vertical velocities and size of the recirculation zones are reduced in the optimized case.

Use of Computational Fluid Dynamics (CFD) in the design process for combustion systems design is becoming an integral part of ABB's design process. For the many types of furnaces and combustor systems developed by ABB, the benefits of using CFD at the design development stage is critical to meet strict operation and performance goals. In addition, CFD is also becoming the method of choice to evaluate potential solutions in case of problems with existing equipment. In the case described here, a vexing corrosion problem inside the combustor of a municipal incinerator was addressed using CFD to model the existing configuration and evaluate alternate solutions. The computer simulation showed that excessive airflow was removing the protective coating inside the heat exchanger, and incomplete combustion left high temperature particles that attacked the tubing. ABB analysts modeled 20 different secondary air arrangements and selected one that was retrofit to reduce the potential for problems. The computer simulation using FLUENT computational fluid dynamics software from Fluent Inc., Lebanon, New Hampshire, provided far more information than was possible using physical model testing and made it possible to evaluate many more redesign options than if testing had been performed. As a result of this and other similar successes, ABB is rapidly moving to incorporate CFD technology in its business units as a product development and trouble-shooting tool.

About two years ago, a customer within ABB reported problems with accelerated wastage and corrosion at specific bare tube regions within the combustor of a municipal incinerator. In normal operation, a protective oxide layer naturally builds up on the fireside of the pressure part waterwall in incinerator systems. However, in some systems high speed air channels compromise the protective oxide layer and thus promote corrosion. This type of problem is actually quite common in municipal incinerators because they generally involve a difficult and ever-changing mix of corrosive chemicals combined with high speed air flow in a reducing environment. Because a minimum level of tube thickness is required for safety codes, this type of problem leads to increased downtime and maintenance. We were called in to help the client determine the source of the problem with their system and then recommend design solutions.

Traditionally, this problem would have been addressed by constructing a cold flow model with airflow scaled to represent the flow patterns. The model would have been built of Plexiglas and smoke tracers would be injected to identify flow patterns. Velocity measurements would be taken with computer-driven pitot tubes. Of course, this model would have to make a considerable number of assumptions. For example, the physical geometry would have to be distorted and the air injectors scaled to represent the rapid expansion of air jets in the combustor using a cold flow model. As a result, the flow and mixing patterns near the jets would differ in important ways from the actual physical situation. Another problem is that cold flow results would be generated only at a few test planes, making it difficult to pinpoint and understand the global flow patterns. Finally, the number of design alternatives that could have been considered would have had to be scaled back since each alternative would have required time-consuming alterations to the physical model.

With these known shortcomings of cold flow model testing, ABB analysts decided to use CFD to simulate the furnace operation. The first task in the simulation was to develop a three-dimensional model to represent the geometry and flow domain of the furnace. The furnace basically consists of a box with a grate at its bottom and wall surfaces formed by a network of adjacent tubes that channel the airflow towards heat exchangers. The CFD model consisted of the combustor itself and the adjacent flues ahead of the heat exchangers.

The next task was to specify the operating conditions, including flow rates for the combustion air and gas coming from the incineration of the fuel. An important issue faced by the analysts was how to characterize the municipal solid waste fuel. It varies as a function of the season and the methods of mixing the fuel. It was approximated in this case by modeling an aggregate gaseous fuel consisting of oxygen, hydrogen and carbon with properties that closely matched the fuel analysis.

The decision to represent the heterogeneous combustion as a gas was made to accelerate the convergence process and increase the number of configurations tested. This simplified the solution procedure. A gaseous fuel was burned using a two step reaction scheme. Of particular interest was the temperature in the region of the secondary air jets. Capturing a reasonable representation of the high gradients experienced by the combustion air jets was an important aspect of the mixing predictions.

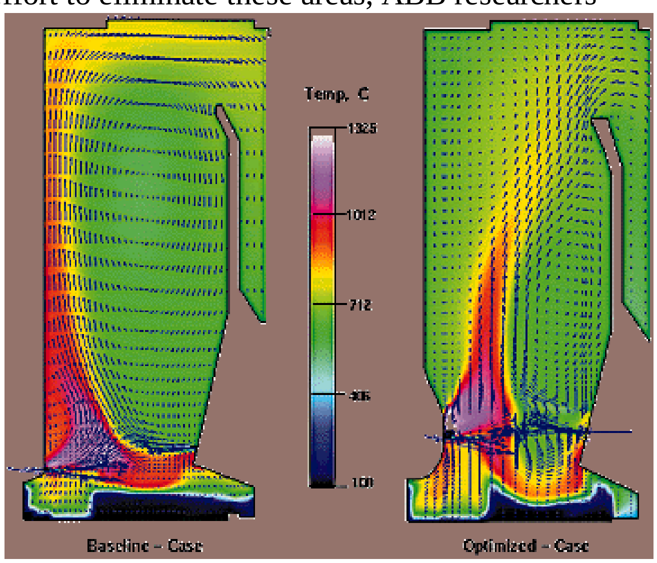
In situations where more rigorous treatment of the fuel combustion is desired, such as close to the grate, solid fuel particles of different size classes would be used to represent the fuel. Fluent software can be used to model solid fuel combustion using a set of flexible process models to represent the physical processes occurring, such as drying, devolatilization, and char combustion. Use of heterogeneous phase models are essential for appropriate treatment of combustion in coal-fired and other types of commercial combustors. Improved submodels currently being incorporated into the software are becoming increasingly more important to generate accurate representations of advanced designs that yield reduced levels of emissions.

After converging the flow field, chemical species, and temperature fields for a particular geometry, analysts compared the results for each case to determine the impact of firing system changes compared to a baseline. Graphical comparisons of the flow and temperature patterns were made. In addition, particle tracers were used to determine the residence time for particles released from different zones on the grate. This provided a relative comparison not only of the residence times but the locations where deposit build-up was likely.

Analysts then turned to validating the results. It is recognized by the combustion community that obtaining direct validation measurements in the form of velocities, temperatures and gas compositions inside an operating incinerator is an extremely difficult task due to harsh environment, size of the furnaces, and lack of access for probing. Therefore, indirect methods are often required to substantiate computer predictions. One technique is to observe the deposit patterns for clues.

Inspection of the furnace revealed the shape of frozen slag deposits on the front wall to have an appearance similar to the crest of a wave. The peak of this wave had projections that were formed as liquid running slag came into contact with the colder air flow. The smooth, curved shape of these deposits clearly revealed the airflow circulation in the immediate region of the jets during operation. The shapes of these deposits correlated well with the peak temperature and air flow patterns predicted by the CFD analysis. Furthermore, ultrasonic testing provided measurements of where the thinning was occurring. These measurements coincided with the regions of elevated velocity and temperature and high particle density in the CFD analysis.

With that evidence to show that the Baseline condition was established, engineers proceeded to evaluating design solutions. The original secondary air configuration was a single level of nozzles. These nozzles inject air into the furnace above the fuel pile, which rests upon a grate. The secondary air has a major impact on the airflow within the furnace because it usually sets up recirculation areas and entrainment zones. These create high velocity airchannels and decrease gas residence time. In an effort to eliminate these areas, ABB researchers evaluated alternative secondary arrangements that included two separate levels of nozzles that varied in the elevation, injection angle, nozzle diameter, and speed at which they enter the furnace. The engineers generated a matrix that included about 20 different configurations.



Carbon-di-oxide distribution is used as an indicator of combustion in the model. The yellow zone near the wall shows the dilution of the burnt gases by the overfire air. The majority of combustion occurs in the front half of the combustor. In this simulation, the optimized overfire air arrangement shifts combustion closer to the grate. This contributes only a minor fraction of the total heat release. This is typical for solid waste combustor systems, which burn a wide range of fuel partical sizes. The rear portion of the grate provides the required time to efficiently burn the remaining combustibles in the ash.

The ideal configuration that the analysts were seeking was a uniform vertical velocity field, with rapid mixing of the secondary air and gas from the incineration on the grate. This minimizes recirculation, particle entrainment zones and high velocity air channels. Improvements made toward achieving these objectives also accelerate particle burn-out before contacting the bare tube walls in the upper region of the combustor.

The analysis showed that the best arrangement consists of two levels of secondary air that are staggered in elevation. Offsetting the height of the nozzles reduces the maximum vertical or lift velocity. The injector angle for each nozzle was also optimized during the analysis. The result was an improved air flow pattern with more uniform mixing, reduced net gas velocity and reduced temperature for the gas entering the convective section. The system was retrofitted according to the results of the analysis about one year ago. Since that time, the rate of corrosion has been substantially reduced.

The traditional method of evaluating design of fossil fuel boilers involves isothermal flow cold models and small pilot scale test furnaces. The test furnaces used at ABB range from the table top models for fundamental property evaluation to a 30 Mw thermal tangentially fired furnace, which is one of the largest experimental furnaces in the world. These experimental facilities are still used for performance testing and to validate computer models. However, ABB Power Plant Laboratories has made ever-increasing use of CFD since it first began experimenting with the technology in the mid 1980s. An important point to note is a 3-way synergy between analytical modeling, cold flow physical modeling, and pilot scale combustion testing. Physical testing helps to validate the CFD analysis while the analysis helps to identify key issues in the physical testing. The main advantage of analysis is that while testing provides information only at the few locations that can be instrumented, analysis provides a global picture of flows and temperatures.

Originally, CFD was applied to cold flow applications, which are simpler because they avoid the complex issues involved in modeling combustion. The use of CFD to model combustion is becoming established, due to recent increases in the software capabilities, faster hardware and increased experience in the user community. The latest trend at ABB is the migration of the technology to the business units. This is a direct consequence of the fact that the technology has been validated to the point that it can be practically applied to real-world customer applications. Business units are using this technology with the goal of reducing emissions and improving combustion efficiency by making evaluations that are too expensive, time-consuming or simply impossible to test. Beside this more R&D oriented application of the CFD technology, the goal is to also integrate it into the normal design process, optimizing the design from the start.