Optimizing the Performance of Ultra-Low-Emission Burners in Refinery and Chemical Plant Furnaces

NPRA Paper # ENV-06-155

Prepared by

Mark Bury, Roberto Pellizzari, and Charles Benson

ENVIRON International Corporation

NPRA Environmental Conference
San Antonio, TX

September 18-19, 2006

Abstract

To satisfy tightening environmental regulations, a new generation of ultra-low-emission burners is being retrofitted to many refinery and chemical plant furnaces. These burners are capable of producing significantly lower NO_x emissions than prior models. However, to obtain optimum performance, careful integration of these burners into a heater's radiant section is critical. If sufficient care is not taken during the retrofit design process, flame impingement on the process tubes may occur. Such impingement will elevate local tube heat flux levels, resulting in decreased run lengths between decoking operations, shortened tube life and, potentially, tube failures. In addition, the NO_x emissions produced by these burners can be significantly influenced by the furnace configuration. Overlooking this impact can lead to emission levels that fall short of expectations.

Computational fluid dynamics (CFD) modeling has proven to be an effective tool in optimizing burner layouts and thereby avoiding furnace performance issues. Consequently, CFD modeling is now being fully integrated into the retrofit design process by major oil companies. These models are able to predict flame shape, heat transfer rates, and tube metal temperatures for proposed retrofit configurations. The likely impact of burner layout on NO_x emissions can also be assessed.

In this paper, the CFD modeling approach for refinery and chemical plant furnaces is described. Practical benefits are illustrated by discussing CFD's role in the development of ultra-low-emission burners and in the successful retrofitting of furnaces.

Background

In response to the emissions constraints imposed by regulatory agencies, many refinery and chemical plant owners are evaluating options to control NO_x emissions from their facilities. Options typically assessed are flue gas clean-up using selective catalytic reduction (SCR) or control of NO_x formation by retrofitting ultra-low-emission (ULE) burners. The latest generation of ULE burners can attain NO_x levels of about 10-25 ppm. Typically the costs associated with retrofitting and operating a furnace with new burners are lower than those associated with the installation of an SCR system. Consequently, burner retrofitting is often the leading candidate in a plant's emissions control strategy.

The outcome of burner retrofit projects can be significantly improved by predicting, at an early stage, furnace performance when operating with the new burners. Design choices, such as burner style, size, number, and arrangement can be optimized through CFD modeling. As a result, problems such as flame or hot gas impingement on radiant process tubes, or flame-to-flame interactions that lead to higher than expected CO or NO_x emissions can be avoided. In addition, changes in the burner layout within the furnace can impact the temperature and composition of the combustion products entrained by the low-emission burners, which can yield lower emissions levels.

To avoid unforeseen performance or emissions problems, major oil companies have begun to include furnace performance modeling as part of the retrofit design process. CFD has been successfully applied in many retrofit projects to guide burner selection and layout within the furnace.

Overview of CFD

CFD modeling involves the solution of the basic equations that describe fluid dynamics, heat transfer, species transport, and chemical reactions for the geometry and boundary conditions of interest. The solution process involves three main steps, namely, problem definition and pre-processing, problem setup and solution, and solution post-processing.

The problem definition and pre-processing step begins with the identification of the questions to be answered by the CFD model. For burner retrofit modeling, prediction of the radiant tube heat flux and temperature profiles is desired, as well as verification that the flame size and orientation are acceptable. The physical domain for burner retrofit cases typically includes the burner geometry, the furnace radiant section, and the first few rows of the convection section. A solid model is built representing the furnace and burner geometry.

The final pre-processing step is the generation of a computational grid for the domain. In this step the domain is broken up into computational cells small enough to capture the relevant flow physics. Careful attention to grid quality is required to ensure a successful solution. Poor grids can lead to numerical instability, or worse, solution results that are not accurate. Grid sizes for these types of models can range from a few hundred thousand computational cells to many millions of cells, depending on the size of the geometry and the complexity of the burners. An example of a typical grid used for a detailed burner simulation is shown in Figure 1 and Figure 2.

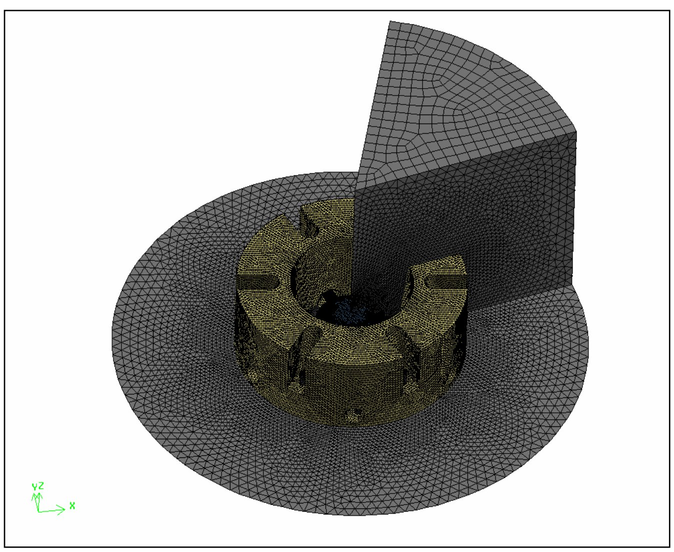

Figure 1: Typical computational mesh for a detailed burner simulation

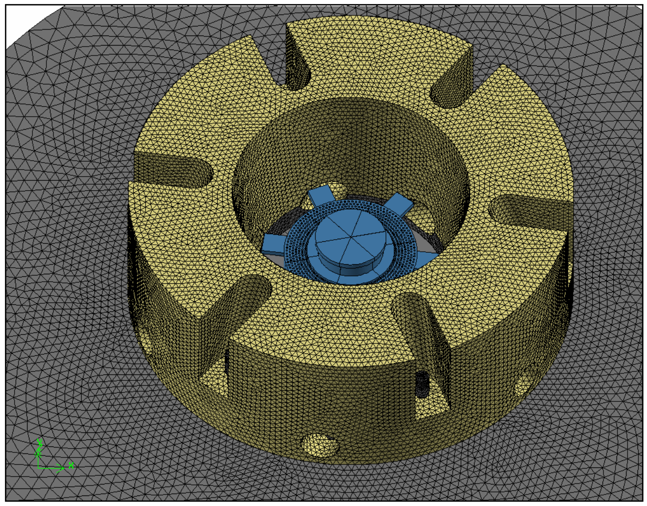

Figure 2: Close up of burner mesh and geometry

Once the pre-processing step has been completed, the grid is imported into a CFD solver. Physical models are selected, boundary conditions (e.g., fuel and air flow rates, radiant tube wall temperatures or heat transfer coefficients, wall emissivity) are specified, and the solution is initialized. The solution of the various equations that describe the physics of the problem (mass continuity, momentum conservation, energy conservation, radiation, species transport, and chemical reactions) is then performed over the domain. It can often take several days for models of large furnaces with many burners to achieve satisfactory convergence.

The final step in a CFD analysis is the post-processing of the solution results. First the solution is examined to ensure that it is physically realistic. Then the solution is interrogated using a combination of contour, vector, streamline, and iso-surface plots. Predictions of the following items are of most relevance:

- Temperature, oxygen, fuel, and carbon monoxide profiles throughout the furnace
- NO_x and CO emissions (trends are predicted more accurately than absolute magnitudes)
- Flame shape and orientation
- Tube wall heat flux and metal temperatures (an example is shown in Figure 3)

Recommendations are then made regarding possible layout or burner modifications. The final step is to then model the modified configuration to confirm that performance improvements are indeed expected. This process is repeated as necessary until a satisfactory configuration is obtained. Experience has demonstrated that this is possible within a few iterations.

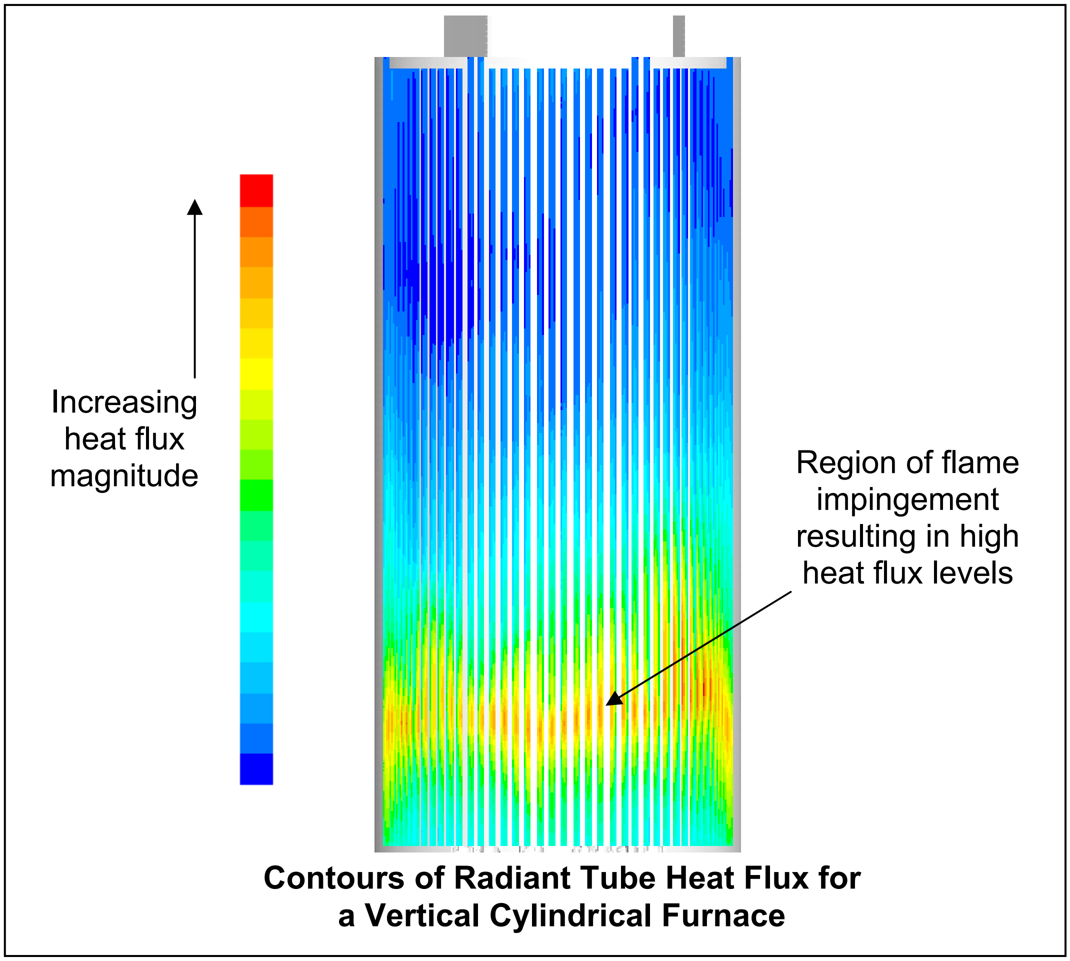

Figure 3: Example of heat flux prediction for radiant section tubes

Ultra-Low-Emissions Burner Development

In the mid-1990's, ENVIRON staff (then working at Arthur D. Little, Inc.) developed a prototype ultra-low-emissions burner[1] to address tightening NO_x emission regulations. The product of this development effort was a 2 MMBtu/hr burner prototype possessing operating and cost characteristics comparable with conventional burner technologies, but having dramatically lower NO_x emissions. Commercially available low-NOx burners of that era typically generated NO_x emissions in the range of 30-40 ppm. The ability of the ULE prototype burner to achieve NO_x emissions of less than 7 ppm was independently verified during testing at Sandia National Laboratory's Burner Engineering Research Facility (BERL).

1. United States Patent #6,007,325

The extremely low NO_x emissions performance of the ULE burner prototype (shown schematically in Figure 4) can be attributed to a combination of design features that include three levels of fuel staging. Each stage has its own distinct combustion zone and dilution mode (via either internal flue gas recirculation or lean premixing). The burner also incorporates a premixed, low emissions continuous duty pilot.

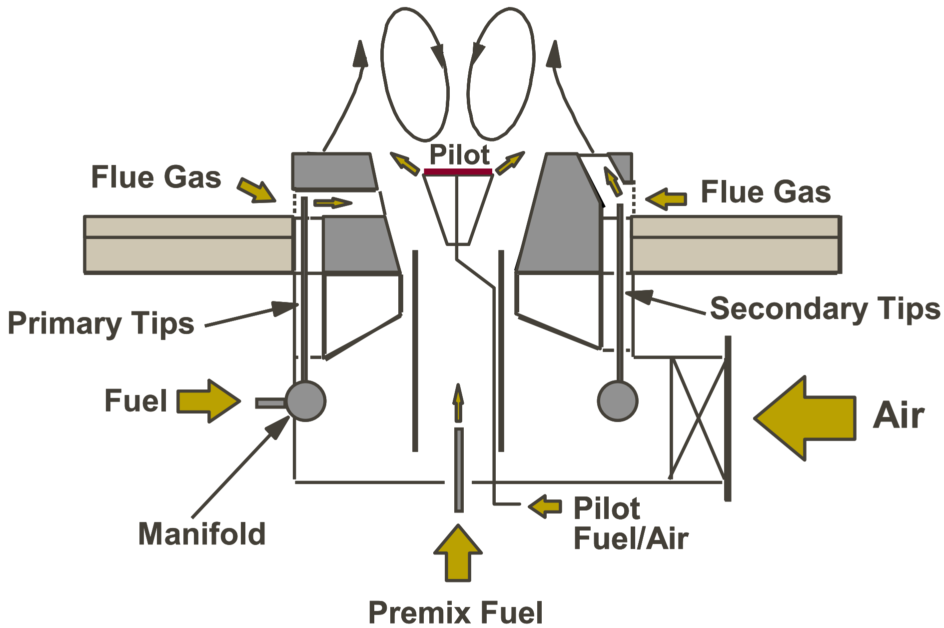

Figure 4: Schematic of ULE burner

Chemical kinetics and computational fluid dynamics modeling played a key role in the development of this burner technology. Coupled chemical kinetics models of the pilot, premixed, primary and secondary combustion zones (a combination of perfectly stirred and plug flow reactor models) were used to optimize fuel distributions, dilution ratios and residence times for both low emissions and stable combustion. Computational fluid dynamics analyses were used to engineer the burner aerodynamics and fuel injection features necessary to approximate the ideal in a practical burner design.

Specifically CFD was used to:

- Design the premix and primary fuel injection features to entrain and uniformly mix sufficient quantities of air and flue gas, respectively.
- Engineer the quarl aerodynamics to achieve good primary fuel/flue gas and air mixing, flame holding behind the continuous pilot and stable combustion.
- Engineer the secondary fuel injection features to achieve the required flue gas entrainment, mixing and location of interception with the main burner flame.
- Locate and minimize the regions of elevated NO_x production.

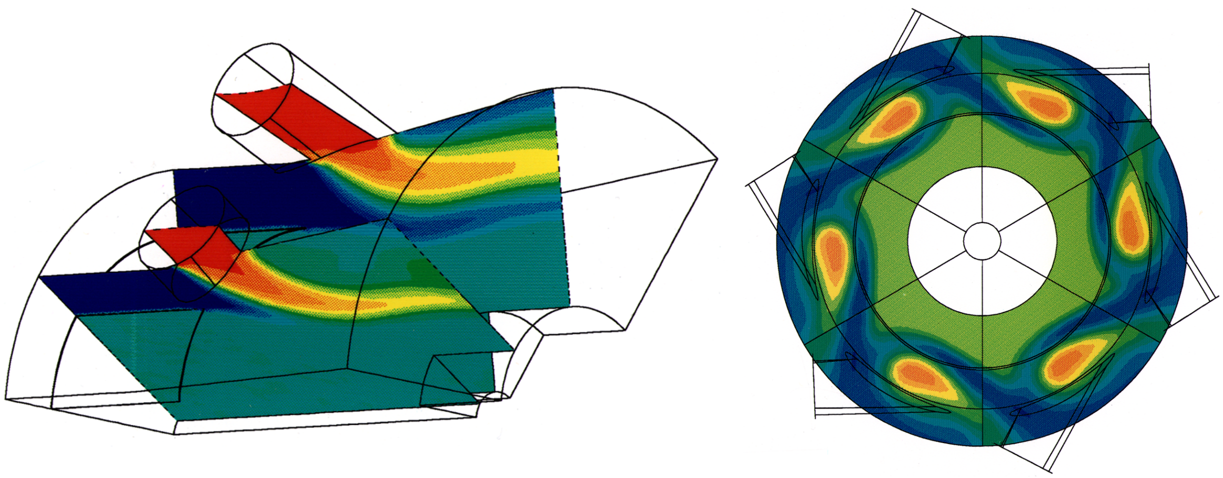

Figure 5: CFD modeling of burner quarl mixing (methane concentrations)

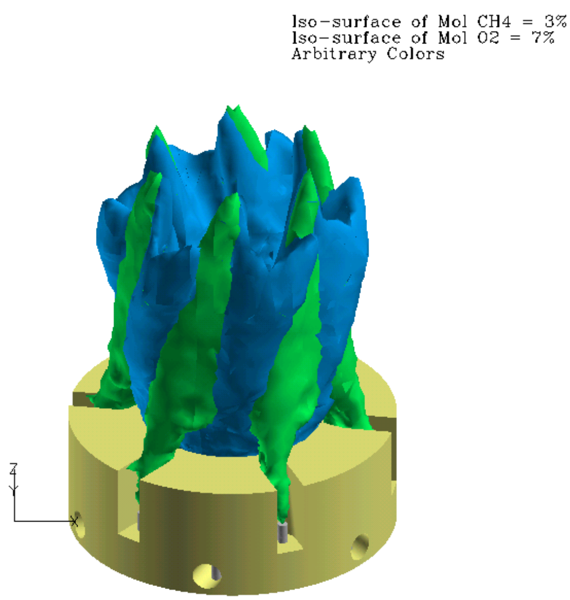

Figure 6: CFD modeling of ULE prototype (secondary fuel injection)

The use of computational fluid dynamics greatly reduced the amount of experimental development effort required to achieve the desired burner performance. Subsequent in-flame measurements of velocity, temperature and species concentrations obtained during independent testing of the ULE prototype by Sandia in their BERL facility (Figure 7) were used to further tune the reaction rate parameters employed in future CFD models of ultra-low-emissions burners.

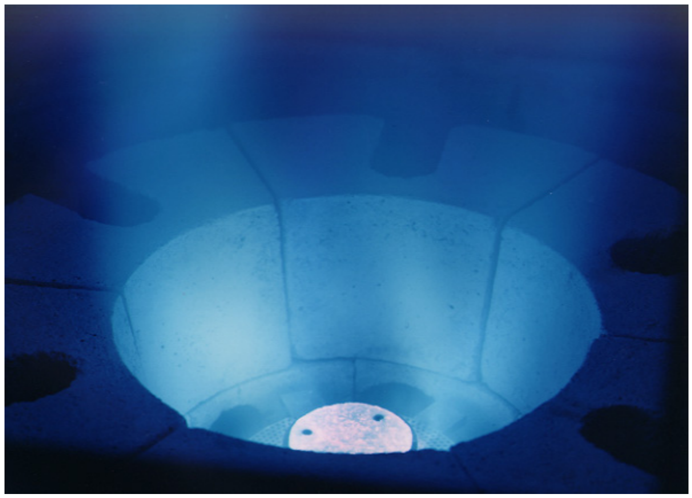

Figure 7: ULE Testing at the BERL

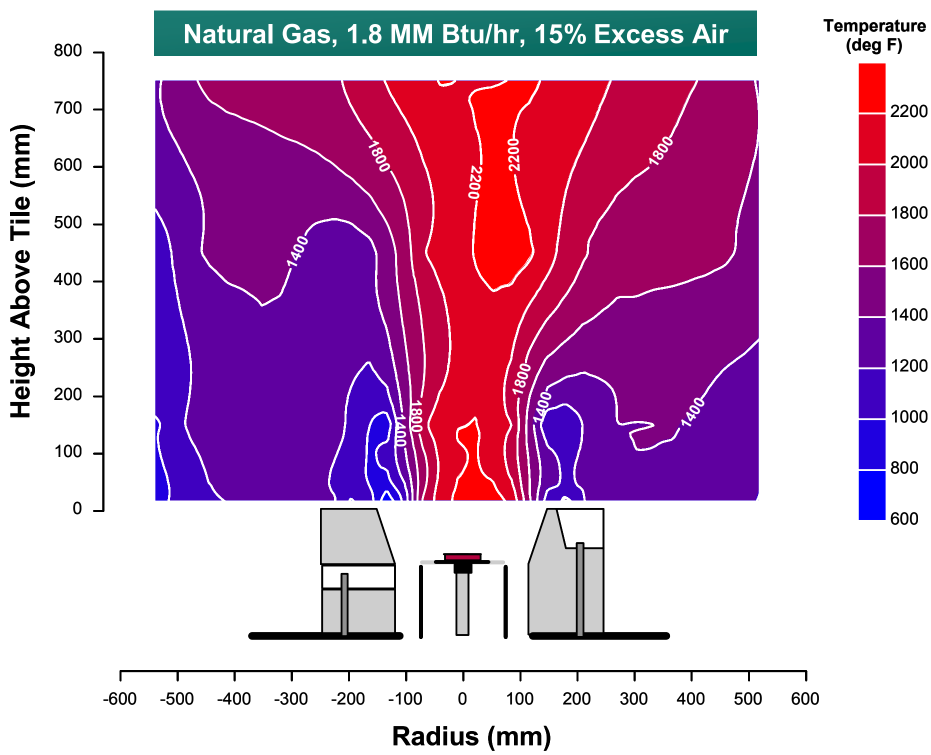

Figure 8: ULE BERL test results (Note absence of high temperature, high NOx production rate regions)

In 2001, a team which included Arthur D. Little, ExxonMobil and Callidus initiated a Department of Energy (DOE) funded project to develop advanced fired heater technologies. A commercial version of the ULE burner developed under this program served as the prototype for Callidus' current Ultra-Blue (CUB) low-NO_x product line.

Fired Heater Retrofit Modeling

A significant benefit of using CFD modeling for furnace retrofits is the ability to identify possible problems before burner hardware is purchased or installed. In addition, working within the constraints of the existing heater structure, the burner layout can be optimized to improve furnace performance. Validation of CFD's use as a risk reduction and performance optimization tool has been realized through many comparisons of model results with field observations and data. CFD has been used to successfully replicate a variety of problems experienced in existing furnaces and to identify solutions that, when implemented, resolved the problems.

The use of CFD modeling to avoid possible performance problems was demonstrated in the DOE Advanced Fired Heater project. An atmospheric pipestill furnace was to be retrofitted with the project's new burner technology. The furnace has a cabin configuration featuring horizontal radiant tubes and 14 floor-mounted, upwardly-firing burners.

The burners generated a swirling flow, driven by primary fuel jets that entrained combustion products delivered into the throat of the burner. In the initial design configuration, identical burners having the same swirl direction (co-swirling) were specified. CFD modeling of the furnace prior to installation of the co-swirling configuration identified a potential flame impingement problem with the end burner (Figure 9).

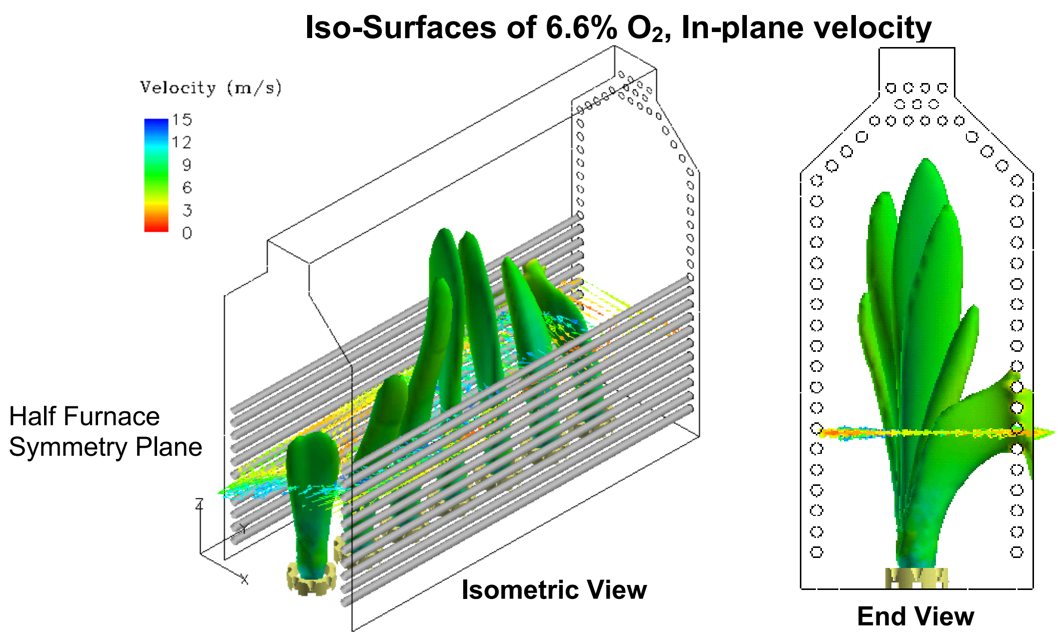

Figure 9: Identification of a potential flame impingement problem

Through analysis of the CFD results it was apparent that the co-swirling configuration generated a swirling flow field around the entire cabin. This flow deflected the end burners into the process tubes as the flow impacted the end wall and turned back along the axis of the furnace. Consequently, a different configuration featuring burners with alternating swirl was proposed. It was envisioned that this configuration would have swirling flow fields that canceled one another.

CFD was again used to model this new alternating swirl configuration. The results showed a significant improvement in the flame straightness and the elimination of flame impingement (Figure 10). Consequently, the alternating swirl configuration was installed into the demonstration furnace. As predicted by the CFD modeling, the alternating swirl configuration produced straight flames (Figure 11).

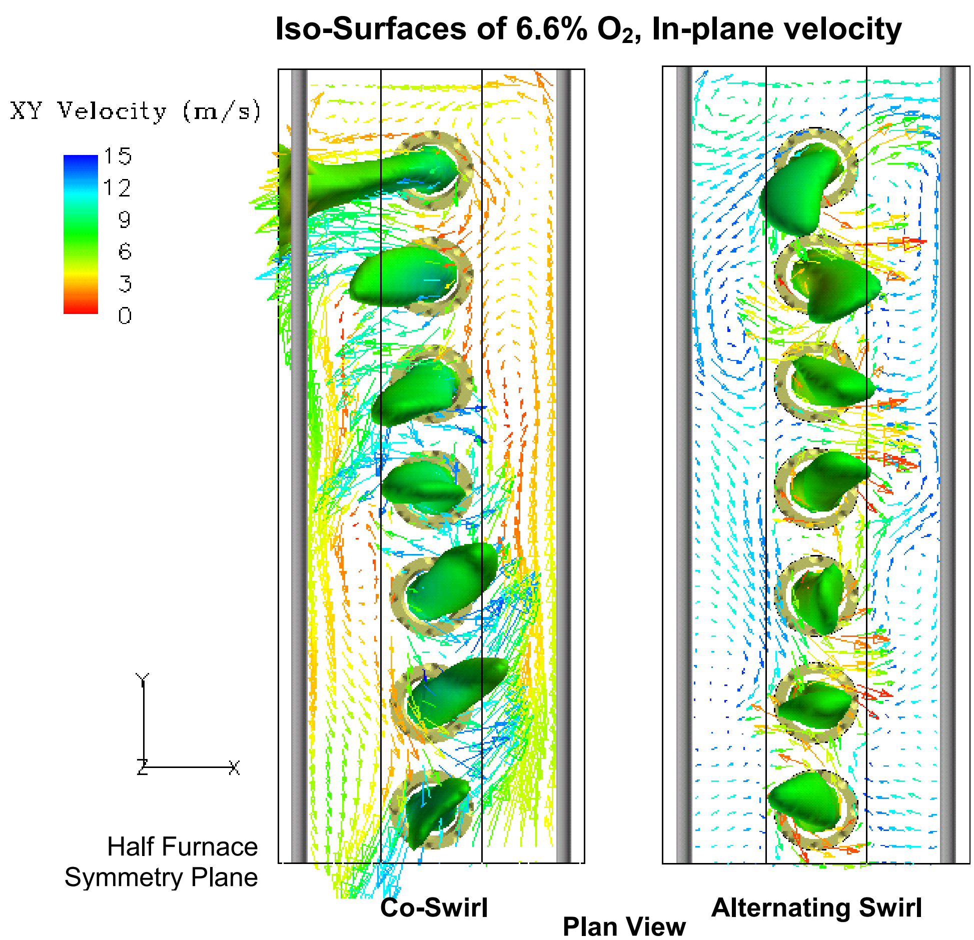

Figure 10: Comparison of co-swirl and alternating swirl configurations

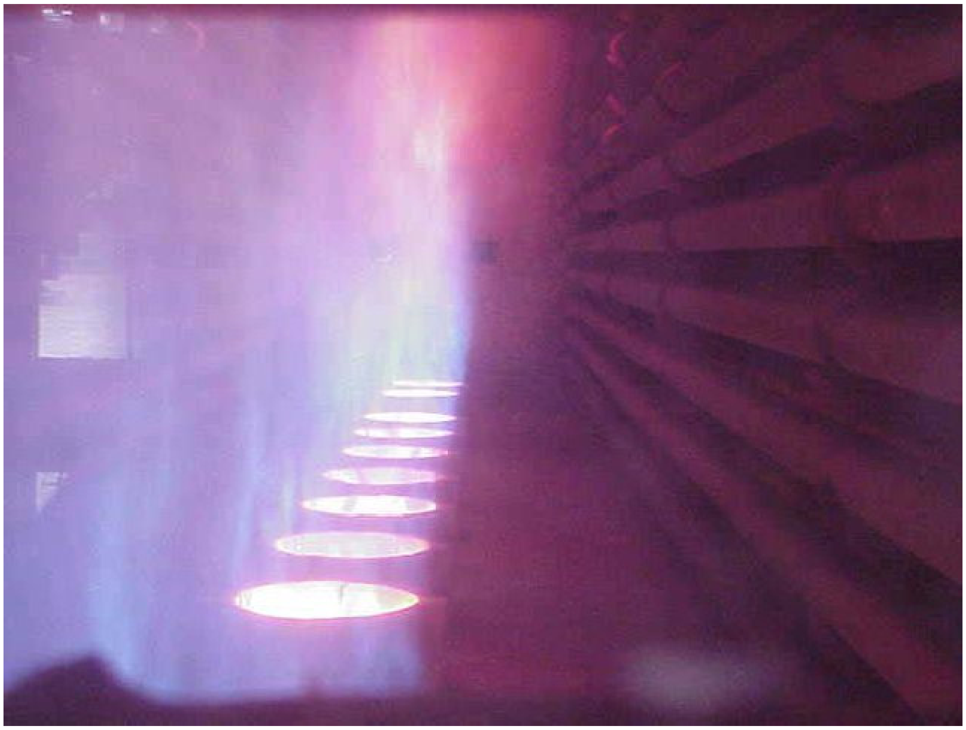

Figure 11: Retrofitted burner in operation

Summary

Retrofitting fired heaters with ultra-low-NO_x burners is an economically attractive approach to achieving regulatory compliance. CFD modeling has been demonstrated to be an extremely useful tool in many successful burner retrofit projects. By applying a proven CFD modeling approach, critical combustion, heat transfer, and emissions-related aspects of burner and furnace performance can be accurately predicted.