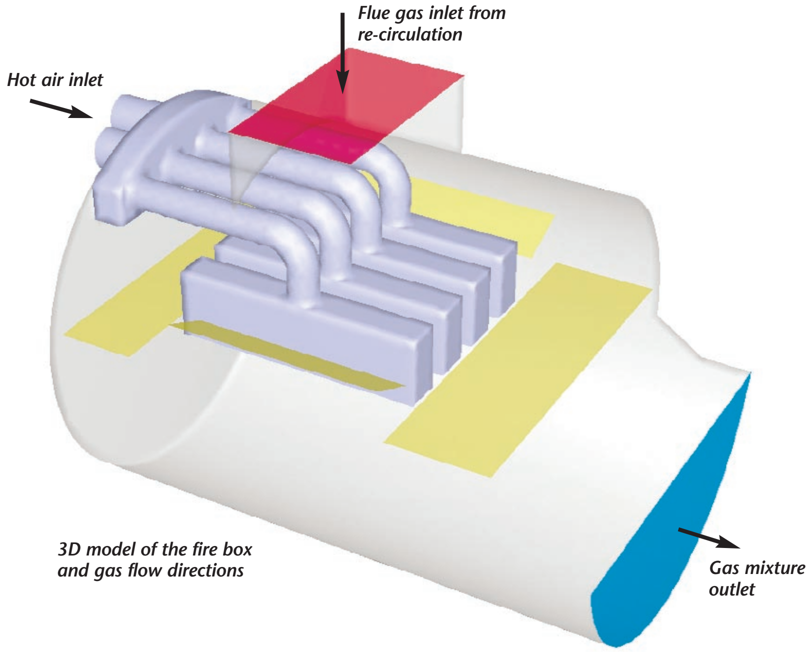
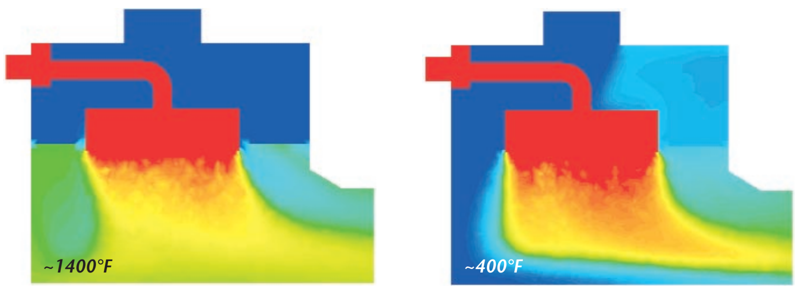


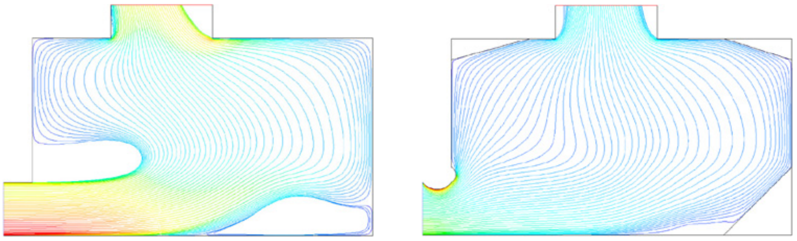
3D model of the fire box and gas flow directions

Troubleshooters Remove **Dead Zones**

By Chang-Ming Lee, Robert A. Curcio, and Gary W. Sams,
Technology and New Product Development, NATCO Group, Inc., Houston, TX



Temperature profiles at cutting planes of X6 with original (left) and modified design (right)



Pathlines colored by velocity magnitude in original (left) and improved (right) inlet design

The Controlled Heat Flux (CHF) heater is an advanced indirect heating system widely used in the oil and gas industry. It uses a forced draft burner where the products of combustion function as the heat transfer medium. The heat generation section (consisting of the burner, or the so-called "fire box") and the heat transfer section (consisting of bundles of heat exchanger tubes with extended surface area) are separated. Heat generated by the burner is transferred to the heat exchangers by the flue gas through forced convection. After passing through the coil bundles, the flue gas moves to the stack, where part of it is recirculated to the burner to lower the flame and flue gas temperatures, and eliminate radiant heat transfer in this section.

Field application engineers discovered two problems with CHF heaters that were in operation. First, they observed that the front shell near the burner section warped after extended use. Second, they observed corrosion on the heat exchanger tubes and fins that was probably caused by a dead zone (or cold spot) in the heat exchanger section closest to the burner. To solve the overheating problem, 3D FLUENT simulations were performed to examine the gas flow and temperature distributions in the fire box section. The geometry based on the interior dimensions included all of the burner components and internal baffles. Rather than use a detailed combustion model, commensurable hot air flow streams with approximate material properties and conformable temperature and pressure values were adapted instead.

The simulations accurately demonstrated the exceptionally high temperature (over 1400°F) observed near the front-end region of the CHF heater vessel. To promote flue gas flow toward the front of the heater a sloped baffle was installed on the top, close to the inlet duct of the flue gas recirculation path. This approach was effective for redirecting the relatively cooler flue gas to the front of the fire box section. As a result, the temperature was reduced to an acceptable 400°F, and the overheating problem was successfully resolved.

A similar simulation approach was used to identify the cause of corrosion in the CHF heater convection coil. A simplified 2D single-phase flow model was employed, and the heat exchanger coil bundles were represented as a porous media with appropriate directional resistance. A pathline plot for the original design illustrated a fairly large dead zone near the inlet boundary of the CFD model. The location of the dead-zone was consistent with the location of the corrosion region (or cold spot) inside the CHF heater. Several attempts were made to alleviate the problem by adding single or dual baffles, both straight and tilted. Ultimately, through a drastically modified inlet design, several internal baffles, altered boundaries for the heat transfer section, and a new location of the stack, the cold spot region was completely eliminated.

By using CFD simulations, NATCO engineers were able to quickly troubleshoot two important problems observed in CHF heaters without the expense of building and testing an actual physical model. The results were accurately validated in later field tests. A multi-million dollar order from a major client was subsequently secured as the result of a similar successful study and dedicated effort. ■