Environmentally Friendly Flameless Furnace

GIULIANO CAMMARATA AND GIUSEPPE PETRONE, UNIVERSITY OF CATANIA, ITALY

We naturally associate flames with fire, but every school child also learns about slowly occurring flameless oxidation such as when iron rusts or wood rots. Meanwhile researchers have learned that even continuous rapid oxidation need not have a visible flame, and a major benefit of furnaces based on flameless combustion is the extremely low amounts of environmentally damaging nitrogen oxides (NOx) they create. In such furnaces, incoming fuel and air must mix with exhaust gases throughout the chamber. Getting the exhaust gases to recirculate to achieve the proper mix is a key to successful operation, so researchers are using multiphysics simulations to help them design and optimize the furnace components responsible for gas recirculation.

NOx also means NOxious

The reduction of NOx has taken on great importance in an environmentally conscious world. NOx contributes to the formation of ozone, it can also cause cardiovascular and respiratory diseases and harm other parts of the body, and it contributes to global warming. Along with sulfur dioxide, it is a major cause of acid rain and can harm the soil.

NOx generation during combustion is typically due to hot spots with high temperature differentials. In a conventional furnace, combustion typically takes place in the range from 1200 to 2500 K, and the gradient between the flame nozzle and other parts of the chamber can reach 700 or 800 K. Yet, while a flameless furnace might have a temperature near 2000 K, the temperature is roughly equal throughout, so no hot spots arise and thus the production of NOx is negligible.

Although reduced NOx emissions are the primary motivation for applying flameless combustion, other benefits include a homogenous temperature distribution throughout the entire combustion chamber and thus less thermal stress on the system for higher reliability, greatly reduced noise (especially important in home furnaces), and fewer restrictions on the types of fuels, because no flame stability is required.

Operating principles

In flameless combustion, air and fuel entering the chamber are mixed with high-temperature recirculated exhaust gases that provide the activation energy needed to initiate and maintain the reaction throughout the chamber. The luminescence of a flame cannot be seen because the reaction does not take place in a specific region, although the primary region of mixing is subject to a higher rate of product formation.

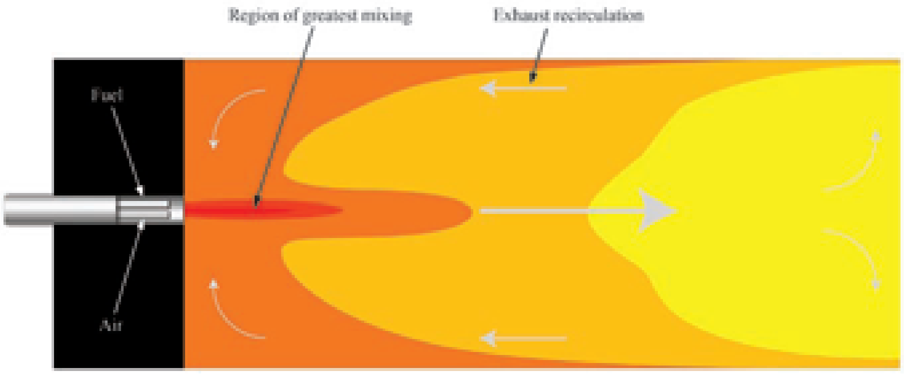

Figure 1: The operation of a flameless-combustion furnace. The recirculation of exhaust gases ensures a flameless combustion and reduces the levels of NOx.

It is important that the fuel and air are injected in such a way that they force recirculation evenly throughout the chamber. This is sometimes achieved with a swirl burner that consists of a series of guided vanes that generate a spiral motion (Figure 2). But with the gases swirling around inside the chamber, what brings them back towards the nozzle area so that they can mix properly with the incoming fuel and air? As the products pass through the swirler at high speeds, their resulting motion generates a low-pressure region inside a spiral field, here called the reverse flow zone (RFZ), that resembles the eye of a hurricane. This low-pressure zone attracts the recirculating gases towards the nozzle to aid in mixing.

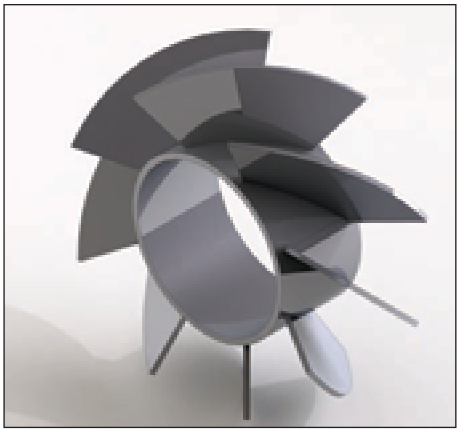

Figure 2: Geometry of the axial swirler.

Simple geometry, complex aerodynamics

Even though a swirl burner might have a relatively simple geometry, the resulting aerodynamics are very complex due to the high level of turbulence. Also note that the spiraling behavior cannot be reproduced by a 2D simulation, so we created one based on fully developed 3D spiral motion.

The first step was to import the geometry of the axial swirler, which was created with SolidWorks® and saved as an IGES file; we brought this drawing into COMSOL Multiphysics with the CAD Import Module. We then divided the geometry

SolidWorks is a registered trademark of SolidWorks Corporation.

Author Biography:

Giuliano Cammarata is a professor of applied thermodynamics and heat transfer at the University of Catania (Italy). From 1994 to 2002 he was Director of the Institute of Technical Physics. He uses CFD tools extensively in his research fields, which include energy distributions in buildings, room acoustics, solar energy, combustion analysis and waste incineration.

Giuseppe Petrone is a researcher in the Department of Industrial and Mechanical Engineering at the University of Catania (Italy). He earned his BSc in Mechanical Engineering at that school and then his Ph.D. at the University of Marne-la-Vallee (France). His primary focus is the use of numerical methods and modeling in fluid dynamics, applied thermodynamics and heat transfer.

into two sections: one dealing with the inlet duct where the swirler is located, and the second representing the initial part of the circular combustion reactor. Then, when meshing the geometry, we used a non-structured mesh made of tetrahedral elements with finer elements close to the swirler zone so we could get more detailed results in that region.

The first step of setting up the model physics dealt with the fluid dynamics of the injection system. The aerodynamics of swirling turbulent jets combine the characteristics of rotating motion and the free turbulence phenomena encountered in jets and wake flows. For this, we used the k-ε turbulence model application mode.

Next, we implemented the oxidation reaction by defining several diffusion-transport equations where the source term drives the chemical kinetics. The maximum value of reaction rate is observed very close to the end of the inlet channels where the combustive material and fuel meet and where oxidation starts. However, the oxidizing agent is present with a high concentration along the entire volume of the cylindrical combustion chamber.

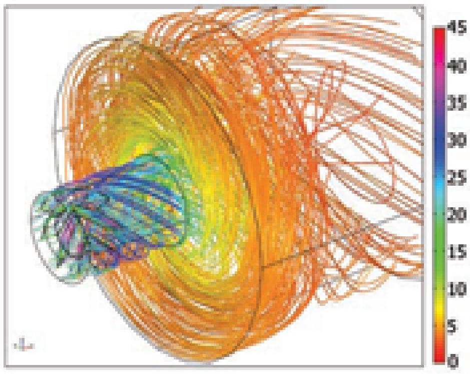

Figure 3: Spiral motion (m/s) imparted to the incoming fluid to spur recirculation.

We then focused on the thermal analysis by using the general heat transfer application mode to relate a source term to the reaction enthalpy and product concentration. Because the gases are extremely hot, they become participating absorbers of radiation, an effect that COMSOL can handle through including expressions in the radiating term in the heat transfer application mode.

In order to estimate heat transfer in the participating media, we assumed that we had an optically thick medium (one through which a photon can travel only a short distance without being absorbed). With such a medium it is possible to express the radiating term as an equivalent diffusive term by introducing a value for global conductivity that takes into account both real conductive and equivalent radiating flux. For the system under study, the radiating properties of the medium satisfy the criterion of an optically thick medium very well, so we could adopt the diffusion approximation in solving the energy equation. The results showed a flat temperature field inside the cylindrical furnace, as expected.

Finally, we solved this nonlinear problem using the UMFPACK solver.

Looking inside the furnace

With COMSOL's imaging capabilities, we were able to see how the fluid accelerates when it moves through the swirler. In addition, when the fluid enters the reactor, it expands. With velocity streamlines we observed the spiral motion imparted to the fluid by the swirler (Figure 3). As noted earlier, the pressure gradients in the spiral core region set up the reverse flow zone, which is clearly visible when you plot the axial velocity isosurfaces as in Figure 4. In that figure, note the bulb located in the central core. It corresponds to the negative values of axial velocity, which means the fluid is recirculated towards the burner outlet. The RFZ is highest close to the outlet and decreases as the fluid reaches the reactor's central zone.

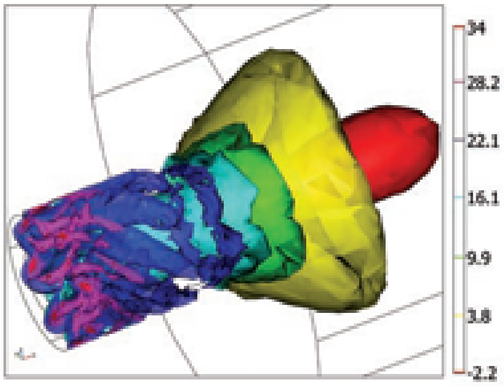

Figure 4: Plot of axial velocity. The bulb in the central core corresponds to the reverse flow zone (RFZ).

The swirler under examination here has proven sufficient for the development of the RFZ, and the recirculation effects reported by the COMSOL model reflect actual swirl-burner behavior. This thermal distribution is in good agreement with experimental data, thereby proving the reliability and effectiveness of the modeling approach. The model is also in good agreement with the literature in terms of both the fluid-dynamic and thermal results. Recall that the main mechanism responsible for the formation of NOx is related to a high temperature gradient during combustion. Because the model shows a temperature field inside the reactor as being almost isothermal, it is safe to assume that NOx formation is significantly hindered.

With this model verified, we can now expand our work to simulate other operating conditions. For example, we plan to perform virtual experiments with other fluids or other inlet velocities without having to conduct expensive experiments. We can also study how the combustion reaction can influence the velocity and pressure fields. ■

READ THE RESEARCH PAPER AT:

comsol.com/industry/papers/1168/