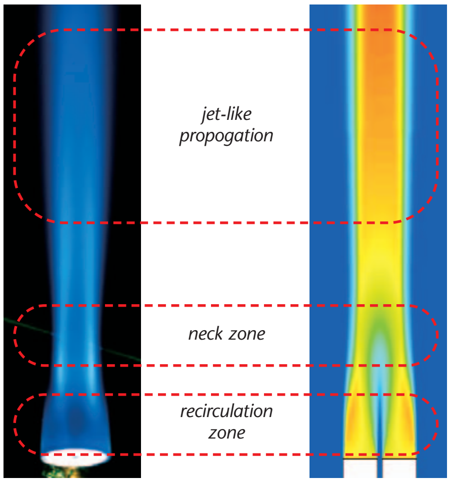
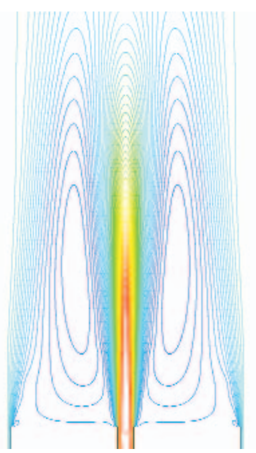
Low Emissions Bluff-body Burner

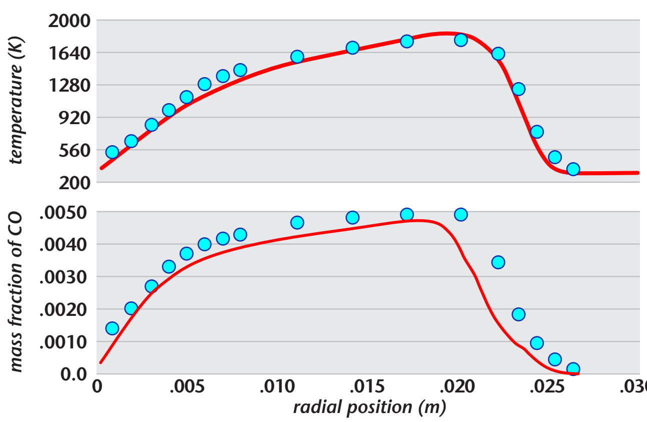
By Antonio Dicorato, Eliseo Covelli, Alessio Frassoldati, Tiziano Faravelli, and Eliseo Ranzi,
Dipartimento di Chimica, Materiali e Ingegneria Chimica, Politecnico di Milano, Italy



Comparison between flame image and temperature field calculated with FLUENT



Contours of axial velocity showing the strucuture of the recirculation zone



Comparison between experimental measurements (points) and model predictions (lines) 30 mm from the inlet

Over the years, CFD has been widely used for understanding and optimizing the design of the combustion process in many systems, such as gas turbines and industrial or domestic heating devices. At Politecnico di Milano, researchers have investigated turbulent non-premixed flames and the environmental impact of pilot burners, with a special focus on flame stability and pollutant formation. In one hydrogen/methane/air flame studied, the presence of hydrogen was found to improve flame stability because higher concentrations of H, O, and OH radicals increase several key reaction rates. The presence of H and OH also result in reduced pollutant emissions. Lean fuel/air ratios lead to a significant shift in flame blow-out conditions, a lower flame temperature, and reduced NOx content. For turbulent jet diffusion flames studied, high jet speed and increased turbulence have been found to increase the mixing and/or ignition of the fuel/air mixture, leading to more stable flames. Turbulence alone, however, is not enough to ensure a stable flame. Recirculation, which can be achieved by inserting a bluff body into an oncoming stream, can be used to increase flame stability even more.

In a recent project, a stable burner with a reduced emissions profile has been studied. The burner is a bluff-body type, which has been presented as a target case at TNF Workshops [1] and for which data are freely accessible on the web [2]. The body produces a low pressure region behind it that entrains gas in a recirculating flow pattern. Fuel is injected into this zone, and is continually re-ignited, sustaining combustion. Behind the recirculation zone there is a neck zone, where the velocity is high and the flame operates close to extinction. Finally, combustion is completed in the flame tail.

The combustion process, including pollutant formation, was modeled using the eddy dissipation concept (EDC) model [3] in FLUENT 6.1. Combustion modeling is challenging because of the strong coupling between the chemical reaction and fluid mixing processes. Often, the rates at which these processes occur differ. Large spatial and temporal variations in species composition and temperature add to the difficulty. The EDC model addresses these issues by simulating the structure of turbulent flames using detailed chemistry, so that the formation of pollutants and other finite-rate effects can be accounted for. According to this model, the chemical reactions are assumed to occur in the fine structures, or small turbulent scales, which are treated as perfectly stirred reactors (PSRs). By treating the reacting fine structures locally as PSRs, a detailed chemical kinetic mechanism can be linked with turbulent combustion modeling. According to the EDC model, chemical reactions are quenched if the characteristic chemical times are longer than the residence time within the fine structures.

To simulate the bluff-body flame, an unstructured, axisymmetric grid of 35,000 cells and a RANS turbulence approach were used. The kinetic model used for methane and hydrogen combustion included 30 species and 110 reactions [4]. Results for temperature, velocity, and composition, particularly CO and CO_2, were found to be in very good agreement with measurements. ■

references:

1 www.ca.sandia.gov/tnf.

2 www.aeromech.usyd.edu.au/thermofluids.

3 B.F. Magnussen, 19th AIAA Aerospace Science Meeting, St. Louis, Missouri, (1981).

4 T. Peeters, PhD thesis, Delft Technical University, Delft, The Netherlands, (1995).