NUMERICAL SIMULATION OF AN INDUSTRIAL PARTIALLY PRE-MIXED RADIANT WALL BURNER

Teresa Guégués[1], Carlos Herce[1] and Cristóbal Cortés[2]

[1]Centre of Research for Energy Resources and Consumption (CIRCE). C/Mariano Esquillor Gómez, 15, 50018 Zaragoza, Spain.

[2]Department of Mechanical Engineering. University of Zaragoza. Campus Río Ebro. Building B. María de Luna s/n, 50018 Zaragoza, Spain.

Introduction

Cracking furnaces are widely used in the petrochemical industry, which continually aims to improve the performance of thermal hydrocarbons cracking. Temperature is a critical parameter in the operation of such furnaces. The wrong operating temperature can result in undesired composition of ethylene and propylene and in failure or rupture of the refractory and coils in the furnace, having an enormous impact on the productivity and profitability. Therefore, modelling the reactive fluid flow in order to understand the heat and mass transfer on industrial scales is very important and has recently become a subject of study. Computational Fluid Dynamics (CFD) has demonstrated to be an efficient tool for combustion simulation. There are very few studies concerned with CFD simulations of steam cracking furnaces at industrial scale. Most of these studies are validated at semi industrial scales and are mainly focused on the behavior of furnaces with long diffusion flames. There is a significant lack of research in partially premixed radiant wall burners at industrial scale.

Additionally, with the increased interest in renewable energies sources and the demand for lower emissions, alternative fuels for power generation have been emerged, such as hydrogen-methane blends. The addition of hydrogen increases the adiabatic flame and decreases the activation energy, resulting in a more efficient thermal process. According to [1], when hydrogen is added to methane, a linear increase of the laminar burning velocity occurs, up to a concentration of X_{H2} of 0.7. However, this has the opposite effect for X_{H2} >0.85, inhibiting methane reactivity. At low hydrogen contents (X_{H2}<0.5), a methane-dominated regime combustion takes place, characterized by a slight increase in the methane laminar burning velocity. Nonetheless, according to [2] the drawback of adding hydrogen is the propensity of auto-ignition or flashback occurrence. Thus, it is necessary to understand the physical and chemical properties of such hybrid fuels, as well as the behavior and stability of the flame.

This work aims to perform a 2D axisymmetric parametric study of the operating parameters of an industrial radiant wall burner. The burner combusts atmospheric air with methane-hydrogen blend (75% CH_4 and 25% H_2). Each air flow is discharged by a high-speed jet from the environment atmosphere, and the fuel at high pressure from a venturi nozzle. The primary air mixes with the fuel but is not sufficient to complete the combustion. Instead, it moves toward the burner tip, where the secondary air is introduced. In the present case, the secondary air entry is geometrically fixed, and the combustion process can, thus, be regulated by controlling the primary air inlet while adjusting the entry diameter between 0 and 9 mm, as well as controlling the amount of air distributed between the primary and secondary entries. The air inlets are represented as simple entries in the grid, but their boundary conditions were calculated in advance according to [3] to account for the pressure loss of the atmospheric air through the conducting tubes. Table 1 shows the some cases simulated by combining the opening of the primary air and the ratio between primary and secondary air.

Numerical Setup

In order to simplify the numerical simulation, the problem was divided into three phases. In the first phase, the non-reactive flow was simulated in order to study the fluid dynamics properties, such as velocity, turbulence, etc. Then, these properties profiles were used as input conditions when introducing chemical kinetics and radiation mechanisms. In the last phase, NOx mechanisms were introduced, since the NOx formation is much slower than the other combustion mechanisms, and thus can be studied after the combustion simulation.

The simulations were performed with a pressure-based solver involving the Reynolds Average Navier-Stokes equations (RANS). The standard k-ε model with a renormalization group (RNG) was used for modelling the turbulence. The radiation model uses a P1 model with a weighted sum of gray gases model (WSGGM). The Eddy Dissipation Concept (EDC) was employed to simulate the combustion-turbulence interaction, and the ISAT algorithm was used to accelerate the solution. In order to take into account the compressible effects, the density was represented with the model of volume-weighted-gases. Several reaction mechanisms presented in literature were compared but the one proposed in *Smooke et al. 1988* [4] was chosen for this study. Although this mechanism was developed for the methane oxidation, it can represent sufficiently well the methane-hydrogen mechanisms with low hydrogen concentration. Hence, as stated in [1], for concentrations X_{H2}<0,5, the methane conversion is dominated by two steps, attributed to the amount H and OH radicals. Those two steps are presented in the Smooke mechanism. The NOx mechanisms considered were characterized thermal and prompt formation. Finally, the mesh was adapted to these simulations through pressure and temperature gradients in order to obtain the mesh independence.

[1] Corresponding author: tguegues@fcirce.com

Name	Pri./Sec. Ratio (%)	Pri. opening (mm)
S1	70/30	9
S2	70/30	3,5
S3	60/40	9
S4	60/40	3,5

Table 1: Combination of the primary/secondary air ratio and primary opening in different simulations.

Preliminary Results

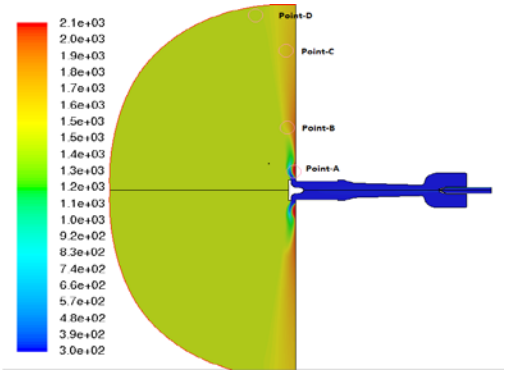

Figure 1: Contours of Temperature in K, and the corresponding locations of the points. Temperature results for simulation S1.

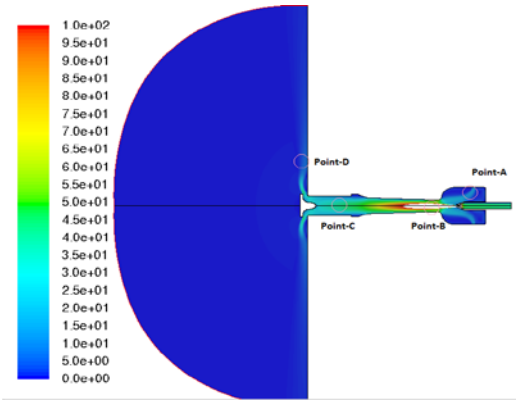

Figure 2: Contours of Velocity in m/s and the corresponding locations of the points. Velocity results for simulation S1.

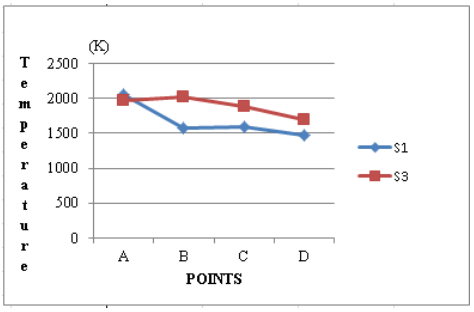

Figure 3: Temperature for points A, B, C and D for different simulations.

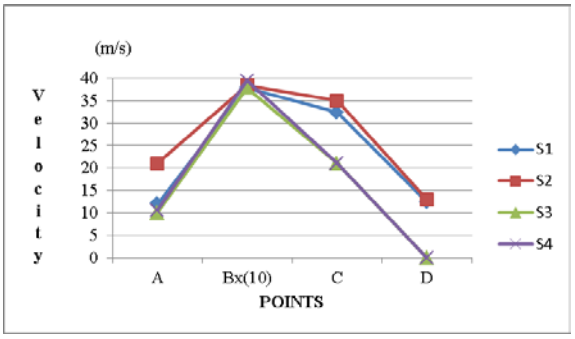

Figure 4: Velocity for points A, B, C and D for different simulations.

Conclusions

Figures 3 and 4 show for the different simulations the temperature and velocity in each point as represented in figure 1 and 2, respectively. Chemical reactions only occur in the combustion chamber after the burner tip. The flame propagates along the wall. However, the flow shows a slight curvature at burner exit, due to the secondary air pressure. One can conclude that with more primary air, the flame temperature tends to be higher and the initial flame curvature is smaller. Variation of the primary air inlet opening diameter resulted in a different flow speed inside the venture and influenced the curvature extension. For the cases with primary air opening diameter of 3,5 mm, the flow moves faster, producing a slight increase in the flame temperature and curvature.

Acknowledgements

This work is supported by the UE Framework Programme for Research and Innovation Horizon 2020 under grant agreement n. 636834 (DISIRE), and the Spanish Ministry of Economy and Competitiveness under grant ENE2013-48003-R (SICOPIE).

References

[1] Sarli, V. Di, Benedetto, A. Int.J. of Hydrogen En., 32: 637-646. (2007).

[2] Morones, A., et al. *Laminar and turbulent flame speeds for natural gas/hydrogen blends*, Proceedings of the ASME Turbo Expo. (2014).

[3] B. B. Daly. *Woods Practical Guide to Fan Engineering*, 6th ed. Woods of Colchester Ltd. (1992).

[4] Smooke, M., et al., Symposium (International) on Combustion 21(1): 1783 –1792, (1988).