Applied Thermal Engineering 31 (2011) 3885–3896

Contents lists available at ScienceDirect

Applied Thermal Engineering

journal homepage: www.elsevier.com/locate/apthermeng

Numerical investigation of burner positioning effects in a multi-burner flameless combustion furnace

B. Danon [a,b,*], E.-S. Cho [a], W. de Jong [a], D.J.E.M. Roekaerts [a,b]

[a] Energy Technology, 3ME Faculty, Delft University of Technology, The Netherlands
[b] Multi-Scale Physics, Faculty of Applied Sciences, Delft University of Technology, The Netherlands

ARTICLE INFO

Article history:
Received 29 May 2011
Accepted 19 July 2011
Available online 26 July 2011

Keywords:
Flameless combustion
Multi-burner furnace
CFD simulation
Burner positioning

ABSTRACT

In this paper results are presented of a numerical study performed for four different burner configurations in a furnace equipped with three pairs of flameless combustion burners firing Dutch natural gas. The simulations have been validated against previously published results of an experimental study [1]. The commercial Computational Fluid Dynamics (CFD) code Fluent 6.3 was used for the calculations. Using the Eddy Dissipation Concept (EDC) model for turbulence–chemistry interaction in combination with the realizable $k-\varepsilon$ model for turbulence and a skeletal chemistry mechanism, the main furnace performance was consistently reproduced for all the investigated burner configurations. Moreover, it was found that due to relatively low Reynolds numbers in the cooling air flow in the annulus of the cooling tubes, predictions of the heat extraction rates of these cooling tubes were improved by treating the flow in the cooling tubes as laminar. Furthermore, the applied error tolerance of the ISAT procedure was insufficient for accurate species concentration predictions, however, based on analysis of the main species concentrations in the flue gas, this inaccuracy did not influence the overall predictions.

The most important experimental results have been investigated using the CFD simulations. Firstly, a longer path length from the firing burners to the stack, compared to the path length to the regenerating burners, explained the lower CO emissions in the flue gas in the stack. Secondly, it was found that a recirculation zone between the upper firing burners and the stack in configurations C4 and C5 resulted in a smaller fraction of the flue gases leaving the furnace via the stack compared to the other configurations. Thus, a larger fraction left the furnace via the regenerating burners and this resulted in higher preheat temperatures of the combustion air. Furthermore, more pronounced recirculation zones in configurations C3 and C4 led to higher temperature uniformities in the furnace. Finally, it was confirmed that the jets of the burners in configurations C1 and C3 showed similar merging behavior, leading to similar NO emissions, as observed in the experiments.

1. Introduction

The industrial wish for higher energy efficiency of large-scale furnaces, and its associated fuel savings, has demanded for new combustion technologies, combining heat recirculation from the flue gas with low pollutant emissions. Flameless combustion (also known as flameless oxidation (FLOX) [2], HiTAC [3] or MILD combustion [4]) is such a novel combustion technique. In this technique the combustion air can be highly preheated, without increasing the pollutant emissions, in particular NO_x. The air and fuel are injected at high velocity and spatially separated. Due to this high momentum injection, large quantities of hot flue gases are entrained into the jets before they mix with each other, lowering the oxygen availability in the reaction zone, thus, also lowering the local reaction rates and the peak temperatures. These low peak temperatures reduce the thermal NO emissions [5].

Since the introduction of flameless combustion in the early nineties of the last century, many universities and research departments of industry have made efforts in experimentally investigating this new technology. These studies have been performed on many different scales, from single open flames in jet-in-hot-coflow setups [6,7] to (semi-)industrial scale test furnaces equipped with multiple burners [8–11]. A focus is here on furnaces equipped with multiple flameless combustion burners.

Many of these experimental studies have been complemented with Computational Fluid Dynamics (CFD) simulations. In the

* Corresponding author. Energy Technology, 3ME Faculty, Delft University of Technology, The Netherlands.
E-mail address: b.danon@tudelft.nl (B. Danon).

1359-4311/$ – see front matter © 2011 Elsevier Ltd. All rights reserved.
doi:10.1016/j.applthermaleng.2011.07.036

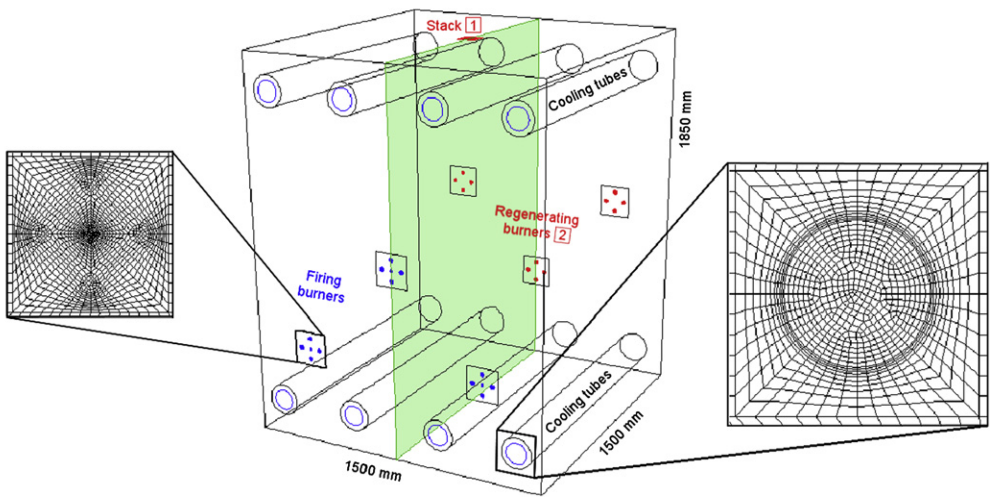

Fig. 1. Furnace sketch, representing burner configuration C5 firing in parallel mode. The boxed numbers 1 and 2 indicate the two sample positions for the flue gas. Sampling point 2 is after the regenerators. The vertical symmetry plane is indicated by the (green) shaded plane. All dimensions are in mm. The two inserts show enlarged front views of the mesh around a burner (left hand side) and a cooling tube (right hand side). The total mesh contains approximately 1.5 million hexahedral cells. (For interpretation of the references to colour in this figure legend, the reader is referred to the web version of this article.)

following the quality of these simulations of furnaces with multiple flameless combustion burners and the choice of physical models in these simulations are discussed.

The 200 kW_th semi-industrial furnace at the Royal Institute of Technology (KTH) has been simulated extensively using the STAR-CD CFD package [12]. Besides the standard $k-\varepsilon$ turbulence model, several combustion models have been investigated, of which the combination of the Eddy Dissipation model and Finite Rate chemistry (ED/FR) turned out to give better results. A two-step chemical mechanism for the combustion of the fuel (LPG, i.e., propane), with CO as an intermediate, was used. Thermal radiation was calculated with the Discrete Transfer method and the absorption coefficient of the gases using the Weighted Sum of Gray Gases (WSGG) model. The cooling tubes were not incorporated in the simulation, but a temperature, based on measured temperatures, was set as a boundary condition. For the validation of the simulations measured heat fluxes, wall temperatures and in-furnace species concentrations were used [13]. The validation results were reasonable, however, the in-furnace temperatures, which this set of models is known to overpredict, were not compared. The three experimentally investigated firing modes, i.e., which combination of burners form a burner pair, were also investigated numerically. The main differences observed in the experiments were reproduced by the simulations. Initially, the analysis of the results was focused on the fundamental properties of flameless combustion. In a later study (firing natural gas) also the observed and reproduced differences between the firing modes were investigated numerically in more detail [14].

In 2004 Hekkens et al. simulated the experiments performed with the 1000 kW_th IFRF furnace firing natural gas using the Fluent CFD package [15,16]. Again, the standard $k-\varepsilon$ turbulence model was used. Three different combustion models were applied; two Probability Density Function (PDF) methods, assuming chemical equilibrium and laminar flamelets, and the ED/FR model. In this last model a two-step chemistry for the combustion of methane was used. Radiation was incorporated using the Discrete Ordinates method and the absorption coefficient was calculated with the domain-based WSGG model. After detailed comparison of the numerical and experimental results, it turned out that the ED/FR model performed best regarding the species concentration predictions inside the furnace. The PDF methods were unable to correctly predict the temperatures inside the furnace, which can be explained by the fact that they assume (too) fast chemistry. However, also the ED/FR model overpredicted the peak temperatures inside the furnace, even though two model constants have been adjusted for flameless combustion purposes using previous IFRF measurements.

The 200 kW_th natural gas fired flameless combustion furnace at the Faculté Polytechnique de Mons has been simulated by Lupant et al. using the Fluent CFD package [10,17,18]. The standard $k-\varepsilon$ turbulence model is applied, with a model constant adjusted for improved prediction of the spreading rate of the jets. Combustion modeling is done using both the PDF method assuming chemical equilibrium and the combined ED/FR model. Again, as in the simulations for the IFRF furnace, two model constants in the ED/FR model were changed in order to improve the results. A one-step

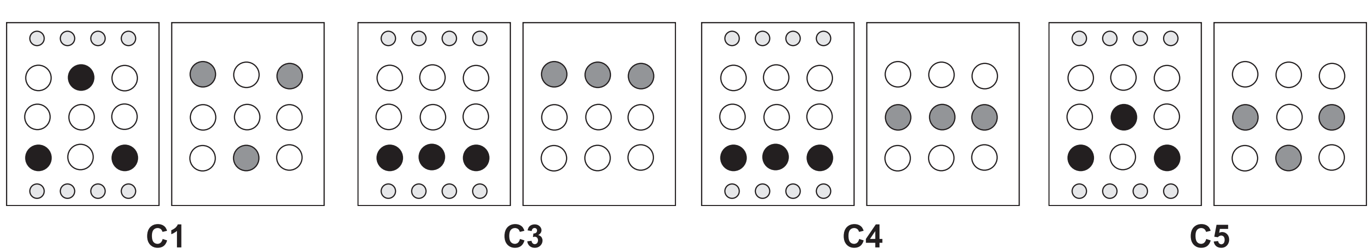

Fig. 2. Overview of simulated burner configurations. The two rectangles represent the two side walls of the furnace with the burner flanges. The large circles represent the burner flanges, if filled black it is occupied by a firing burner, if filled gray it is occupied by a regenerating burner. The black and gray circles switch after a period of 30 s. The small circles denote the position of the cooling tubes.