Applied Thermal Engineering 105 (2016) 537–548

Contents lists available at ScienceDirect

Applied Thermal Engineering

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

Research Paper

Effects of excess air and preheating on the flow pattern and efficiency of the radiative section of a fired heater

Erfan Khodabandeh [a], Mahdi Pourramezan [b,*], Mohammad Hossein Pakravan [c]

[a] Department of Mechanical Engineering, Amirkabir University of Technology, Tehran, Iran
[b] Department of Mechanical Engineering, Faculty of Engineering, Ferdowsi University of Mashhad, Mashhad, Iran
[c] Department of Mechanical Engineering, University of Dayton, Dayton, United States

HIGHLIGHTS

- A box-type fired heater with swirling bluff-body burners is studied by CFD.
- RSM along with DO model can more precisely predict the flow and temperature.
- Multiple vortices are formed, and the flow is complex and asymmetric.
- IRZ and ERZ are affected by variation of excess air and preheating temperature.
- Preheating increases the efficiency, but its effect is reduced at high temperatures.

ARTICLE INFO

Article history:
Received 29 December 2015
Accepted 5 March 2016
Available online 21 March 2016

Keywords:
CFD modeling
Fired heater
Bluff-body burner
Flow pattern
Efficiency

ABSTRACT

Since boilers and industrial furnaces are one of the largest consumers of fossil fuels, optimization and retrofitting these facilities are of the most essential tasks which can be done to reduce fuel consumption and pollutant emission into the environment. The objective of this paper is to investigate the effects of excess air and inlet air preheating on temperature, flow, and efficiency of the furnace of a fired heater used in an oil refinery. For this purpose, a three-dimensional modeling was performed, using CFD, to simulate the furnace and one of the swirling bluff-body burners. Examining the result of several turbulence and radiation models, it was found that Reynolds stress model (RSM) can simulate turbulence caused by swirling and bluff-body burners more precisely than k–ε models. Moreover, discrete ordinate (DO) model offers better performance than P1, for radiation. Comparisons between the numerical results, and field measurements showed the simulation can predict the flow and temperature fields in the heater with acceptable accuracy. The results show that internal and external recirculation zones (IRZ & ERZ) are caused by swirling and bluff-body flows, respectively. In addition, the flow is asymmetric and two recirculation zones form near the vertical walls. Variation of excess air has more impact on IRZ, and high percentage of excess air leads to narrower IRZ, while preheating totally affects ERZ. The optimal value of excess air is 18% and increasing excess air beyond 18% reduces the flame temperature and the furnace efficiency. Furthermore, it is found that preheating increases the furnace temperature as well as the efficiency, and the effect of preheating is reduced at high inlet temperatures.

1. Introduction

Although the widespread applications of renewable energy resources have drawn the attention of many engineers and investors, there is still a great demand for conventional fuels all around the world [1]. This demand, which results in the rising prices, in conjunction with the limited resources of these fuels, and also environmental considerations, have led into the numerous investigations on the enhancement of the working characteristics, along with the reduction of fuel consumption of energy systems in the industrial sector of many countries [2–4].

Refineries, as one of the most important parts of petrochemical industry, are places at which several kinds of boilers, furnaces, and heaters can be found. At refineries, in order to convert crude oil into different products, a number of specific processes have to be conducted. Such processes either have more efficiency at high temperatures or are impossible to be carried out at low temperatures.

* Corresponding author. Tel.: +98 9153219376; fax: +98 5132768836.
E-mail address: m.pourramezan@stu.um.ac.ir (M. Pourramezan).

http://dx.doi.org/10.1016/j.applthermaleng.2016.03.038
1359-4311/

Nomenclature

$D_{L,j}$	molecular diffusion
$D_{T,j}$	turbulent diffusion
F_j	production by system rotation
G_j	buoyancy production
g	gravitational acceleration vector (m/s^2)
P_j	stress production
Pr	turbulent Prandtl number for energy (=0.85)
p	the fluctuation of static pressure (Pa)
S_{user}	user-defined source term
T	temperature (K)
t	time (s)
u'	fluctuating velocity component (m/s)
x	Cartesian space coordinates (m)
Greek symbols	
β	coefficient of thermal expansion
ε_j	dissipation
μ	turbulent viscosity, kg/m s or Pa s
ρ	fluid density, kg/m^3
ϕ_j	pressure strain
Ω_k	mean rate-of-rotation tensor

Therefore, there is a need for a device, which can raise the process fluid temperature. To meet this need, fired heaters (or process heaters) are utilized in refineries. A fired heater is a direct-fired heat exchanger, which uses the hot gases of combustion to raise the temperature of a feed flowing through coils of tubes aligned throughout the heater. These devices usually increase the temperature of the process fluid (crude oil) approximately from ambient temperature (≈300 K) to the desired one (≈400 K), at which delivered to the distillation unit. The required heat of this process is provided by combustion, which is a complicated phenomenon with a variety of applications in different industries. One of the chief concerns in designing combustion systems is to control the dimension and stability of the flame. For this purpose, two main approaches have been proposed: (1) the mechanical methods, such as using the flame stabilizer by inserting solid obstacles in the stream, as in ramjet technology (bluff-body stabilization). (2) Hydrodynamic methods, such as swirling by directing part of the flow or one of the flow constituents, usually air, opposed or normal to the main stream, as in gas turbine combustion chambers (aerodynamic stabilization) [5].

Investigating the abilities and effects of these methods on the burners and flame characteristics, several experimental studies have been conducted in various applications [6–10]. Recently, Herrera et al. [11] studied the application of a porous burner made of a bed of low cost Al_2O_3 particles coming from grinding wastes and combined with a ceramic sponge of SiSiC for cooking in the food industry. They stated that the improvement in efficiency and the possibility of interrupting the flow of fuel in a cyclic operation are highlighted as important characteristics for development of more efficient and less-fuel consuming devices for cooking in the food industry. Using twisted tapes of different twist ratio, Hindasageri et al. [12] studied the effect of swirl on flame jet impingement heat transfer characteristics. They found that swirl enhances the heat flux distribution by 40–140% at low Reynolds. However, at higher Reynolds the effect of swirl was negative and was found to decrease the average heat flux distribution by 10–40%.

Due to the difficulties and obstacles associated with experimental studies on the combustion devices, the application of Computational Fluid Dynamics (CFD) has become a useful tool to predict the performance and operational conditions of these facilities. Therefore, numerical modeling of burners and furnaces has been the focus of several studies in recent years [13–18].

Syred and Beér [19] categorized the swirl combustors into two main types; the swirl burner, and the cyclone combustion chamber. They stated that swirl increases considerably the stability limits of most flames, and contrary to many previous assumptions, the flow is often not axisymmetric but three-dimensional time-dependent. Using a finite difference modeling, Lilley [20] investigated turbulent boundary layer of swirling flames, and demonstrated that the effects of swirl on flame size, shape, and combustion intensity are represented by nonisotropic mixing-length and energy turbulence models and an eddy-break-up turbulence-controlled reaction model. Utilizing PDF (probability density function) combustion model and RSM (Reynolds Stress Model) for turbulence, Meier et al. [21] performed a full-scale simulation on a 150 kW furnace and compared the results with experimental data. Later, based on the results obtained from CFD computations, Frassoldati et al. [22] focused on a new procedure for the determination of NO_x emission from combustion processes, which allows using very detailed and comprehensive reaction schemes. Testing the predictions on flames with different swirl numbers, they reported that both CFD, and the chemical analysis show a satisfactory agreement with the measured data. Furthermore, studying the effects of the thermal radiation model on combustion, Ilbas [23] simulated a turbulent non-premixed flame with and without radiation models. They observed that the results with radiation models are in better agreement with the measurements, compared to the results without any radiation models. In addition, the use of a radiation model leads to predict lower overall temperature, and consequently, lower NO_x emissions.

Besides the works mentioned previously, the process heaters have been specifically considered by researchers. Most of these works approached the problem from the energy and environmental points of view, and few of them utilized the numerical method to investigate the heaters. Some instances are studies conducted by Mussati et al. [24], Morales-Fuentes et al. [25], Varghese and Bandyopadhyay [26,27], Chaibakhsh et al. [28], Jegla et al. [29], etc.

The present paper describes the results of a numerical modeling of a fired heater in a refinery. A special attention is exerted to the swirling bluff-body burners and furnace zone, which is the radiative part of the heater. In addition, the effects of excess air and inlet preheating temperature on the flow pattern and performance of the heater are investigated. Comparison between CFD simulation results and those of other experimental works along with temperature measurements have been conducted to achieve a preferable modeling approach, and also to validate the model.

2. Description of the fired heater

Process heaters are generally built with two distinct heating sections: a radiative section, variously called combustion chamber or firebox, and a convective section followed by a stack (see Fig. 1). The tubes in the radiative section may be arranged horizontally or vertically along the heater walls, including the hip, and the burners are located on the floor or on the lower part of the longest side wall, where there are no tubes [30].

Process heaters are usually classified as vertical cylindrical [31] and box-type [28] heaters, depending on the geometrical configuration of the radiative section. Box-type furnaces are best suited for