Applied Thermal Engineering 105 (2016) 876–886

Contents lists available at ScienceDirect

Applied Thermal Engineering

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

Novel approach to proper design of combustion and radiant chambers

Zdeněk Jegla, Bohuslav Kilkovský, Vojtěch Turek *

Institute of Process Engineering, Faculty of Mechanical Engineering, Brno University of Technology, Technická 2, 61669 Brno, Czech Republic

ARTICLE INFO

Article history:
Received 15 December 2015
Revised 16 February 2016
Accepted 13 March 2016
Available online 22 March 2016

Keywords:
Combustion equipment
Heat flux distribution
Thermal loading
Low-emission burners

ABSTRACT

Combustion and radiant chambers with inbuilt heat transfer surfaces are integral parts of a wide range of process and power equipment such as fired heaters, power boilers, or incinerator furnaces. Operating problems that many of these combustion chambers suffer from are typically due to the design procedures using data of insufficient accuracy regarding the calculated local heat transfer data in individual parts of the chambers equipped with modern low-NO_x burners. These problems, obviously, force the designers to devise more accurate design procedures for the respective equipment. The paper therefore discusses the main results obtained so far in the course of a several years long research effort and presents a basic outline of an up-to-date, novel approach to proper design of combustion and radiant chambers with inbuilt heat transfer surfaces. The three most important and – considering the current design practice – also original components of the presented novel approach which significantly improve the quality of the resulting combustion chamber designs are (i) experimental determination of the actual burner heat flux distribution, (ii) determination of the actual fuel burnout profile of the burner from the obtained heat flux profile using the validated MPF model, and (iii) utilisation of the respective fuel burnout profile in the course of design of the combustion chamber and its inbuilt heat transfer surfaces together with calculation of the local heat flux distribution via the plug-flow-based method, thus replacing the currently used design methods based on the "well stirred" models.

1. Introduction

Heat transfer surfaces are commonly present in combustion chambers of many types of process and power equipment – e.g. in waste incinerator furnaces, power boilers, or fired heaters utilized in refineries or petrochemical industry. In such cases it is crucial that the actual heat flux distribution on the embedded heat transfer surfaces is identified as early as in the design stage because it is necessary not only for the purposes of cost estimation but mainly in order to be able to guarantee the required equipment reliability and service life. Even though this is a long-known fact (see e.g. [7]), it still has not been properly implemented into the respective design procedures.

Currently available approaches to this problem are often based on empirical data obtained for older, classic types of burners [11]. However, the wide use of low-NO_x burners both in case of new equipment and for retrofits brings about various operating problems as well as shorter life spans of some of the burner parts. These are then due to a largely different heat flux distributions compared to the ones seen with conventional burner types [21].

Common low-NO_x burners with staged supply of fuel or combustion air into the combustion space exhibit significantly different characteristics (flame length and width) compared to the single stage burners. This is why transfer and distribution of heat from the flame and combustion products to the heat transfer surfaces present in the combustion chamber are different as well should low-NO_x burners be used instead of the conventional ones. The extent of such differences then depends on the actual geometry of the burner and utilized low-emission combustion technology [5].

It is therefore obvious that after replacing a conventional burner with a low-NO_x one changes would be seen not only in the produced emissions but also in thermal-aerodynamic ratios of the entire combustion chamber [5]. These, however, are very often ignored by the operators – especially if the operation had been trouble-free up until the point when the burner was replaced. The usual consequence of such a burner replacement then is the inability to meet the process technology requirements after a short time of operation, often in conjunction with damaged portions of the heat transfer surfaces in the combustion chamber [21]. Specifically, in case of combustion chambers in furnaces operated in gaseous or liquid waste incinerators or in power plants the often-encountered consequences are locally increased thermal

---

* Corresponding author.
E-mail address: turek@fme.vutbr.cz (V. Turek).

http://dx.doi.org/10.1016/j.applthermaleng.2016.03.068
1359-4311/

loading of portions of heat transfer surfaces [33] leading to tube deformations or cracking of the joints of tubes and headers [32]. Considering radiant sections of fired heaters used in refineries or in petrochemical industry, the consequences are often even more severe. Here, due to local overheating of the tube surfaces cracking or coking of the heated hydrocarbon mixture occurs [11]. As a result, not only that quality of the heated mixture is lower but there also is an increase in operating cost caused by larger pressure drops induced by layers of coke on the tube inner surfaces. Such fouling generally warrants a service shutdown and cleaning of the tubes in the radiant section of the respective fired heater [21]. If these fouling layers are not removed then the increased thermal loading together with the layers themselves cause overheating of the tubes and, consequently, degradation of the tube material. The first visible sign of such issues often is observable deformation of some of the tubes in the combustion chamber [19]. If this is not discovered early on then the respective tubes may crack, the hydrocarbon mixture may leak into the combustion chamber, and, ultimately, the fired heater may explode and possibly even destroy other equipment nearby or cause fatalities [21].

It must be emphasized that there is a significant inconsistency in combustion chamber design procedures. On the one hand, chamber dimensions are often estimated via various simplified methods which do not take into account the actual heat flux distribution caused by using modern burners and thus the aforementioned operating problems are usually encountered. On the other hand, sophisticated and detailed computational tools (typically based on computational fluid dynamics – CFD) are employed but these, given their demanding nature, are ordinarily the methods of choice only when an existing apparatus is being analysed. One is therefore able to identify the source of the operating problems (non-uniform thermal loading of the heat transfer surfaces) but, as confirmed by the review of the state of the art further in the text, no remedial actions are offered by such methods.

A specific way to overcome this design-operating inconsistency considering combustion equipment is discussed in this paper. The authors present a novel method based on their long-term experience which identifies the actual thermal properties of modern, low-NO_x burners and then takes the obtained data into account during preliminary design of the combustion chamber and dimensioning of the inbuilt heat transfer surfaces. Possible operating problems due to uneven thermal loading are thus effectively abated (if not eliminated altogether) right from the start.

1.1. Conventional modelling approach vs heat flux distribution modelling

Modelling of creation and growth of the fouling layer inside the radiant tubes of process fired heaters due to coking of the hydrocarbon mixture is discussed in a vast number of papers; however, the crucial factor of heat flux non-uniformity is not considered sufficiently in them. Souza et al. [36], for example, presented an axisymmetric, two-dimensional CFD model for prediction of coke formation inside the tubes of refinery fired heaters under constant thermal loading due to thermal cracking of heavy petroleum residue coming from a vacuum distillation column. Cross [11], on the other hand, presented a very simple computational tool for prediction of coking of hydrocarbon mixtures inside fired heater tubes which was based on the Nelson thermal decomposition constant, that is, in his model the predicted coking tendency was temperature-dependent. Together with using a simple thermal model of a radiant section of a fired heater which splits the section vertically into a lower, flame zone and an upper, flue gas zone, it clearly demonstrates how significantly the accuracy increases if heat flux distribution is considered while evaluating the fired heater thermal performance and the effect of coking.

Morales-Fuentes et al. [30] introduced a simple deposition-removal model which predicts fouling rates and its effect on the thermal–hydraulic performance of a fired heater under constant heat fluxes in radiant and convection sections of the heater obtained from the heater thermal balance. The model allows for prediction of the increase in fuel consumption as the thermal resistance builds up on the heat transfer surfaces and also production losses being a consequence of increased pressure drop.

Later on, Morales-Fuentes et al. [29] applied this simple model to prediction of local fouling rates while utilizing thermal–hydraulic data obtained for the heater with a commercial software FRNC-5PC in which the radiant chamber was simulated using the approximate “well stirred” furnace model (i.e., under the assumption that the flue gas temperature in the chamber was constant and, consequently, that the heat flux was constant as well). The effects of excess air and steam injection were also studied as a means to mitigate fouling.

Chaibakhsh et al. [8] presented a dynamic fired heater model intended for long-term simulations of crude oil direct fired heater operations. The effects of flame height as a function of burner duty was included via the simple Heskestad relation.

1.2. State of the art in heat flux distribution modelling

In contrast to the methods mentioned so far, generally applicable ways of identification of the actual heat flux distribution in industrial combustion chambers are discussed significantly less often – typically only as the necessary introduction to some model developed for a specific industrial combustion chamber and its specific operating problems.

Given the complexity of the issue and the variability of combustion chamber designs used in process and power equipment, currently the prevalent ways to analyse thermal loading are either via modern, detailed, and often also computationally very demanding CFD models or via simple empirical models for conceptual or basic design of the respective chambers.

As for the first category, the recent efforts include e.g. the paper by Pan et al. [34] which presents an interesting thermal–hydraulic analysis of a combustion chamber containing membrane waterwalls installed in a 600 MW supercritical fluidized bed coal-fired boiler. Gandhi et al. [12] predicted using a detailed CFD analysis the ash flue gas erosion rate in an operated large-scale wall-fired pulverised coal boiler in order to overcome different issues such as slagging, fouling, or corrosion–erosion which can all damage the boiler in the long run. Moghari et al. [28] presented their numerical study aimed at determining the temperature distribution of water and flue gas flows in a D-type water-cooled natural gas-fired boiler and therefore also at obtaining an accurate assessment of the respective thermal performance. Ludowski et al. [26] carried out a CFD simulation for a three pass platen superheater placed in the combustion chamber of a circulation-fluidized bed boiler burning coal or biomass and operated for several years which was aimed at improving the superheater design and operation. Jegla et al. [19] presented the results of numerical parametric analyses of the effect of common types of radiant tube deformations on heat flux non-uniformity encountered in a vertical cylindrical 24 MW fired heater operated in the Czech Republic. A year later, Jegla et al. [22] carried out a CFD simulation of the radiant chamber in this heater and provided the obtained results in a form suitable for furnace designers evaluating circumferential and longitudinal heat flux variability factors of low-NO_x burners operated therein. Adamczyk et al. [1] investigated the impact of various air- and oxy-fuel operating conditions on temperature, pressure, concentration, and burn-out profiles within a large scale industrial circulation-fluidized bed boiler installed in Poland and included both experimental and numerical data in their paper. Last but