CHEMICAL ENGINEERING TRANSACTIONS

VOL. 35, 2013

A publication of AIDIC
The Italian Association of Chemical Engineering
www.aidic.it/cet

Guest Editors: Petar Varbanov, Jiří Klemeš, Panos Seferlis, Athanasios I. Papadopoulos, Spyros Voutetakis

ISBN 978-88-95608-26-6; **ISSN** 1974-9791

DOI: 10.3303/CET1335199

Development of Modified Plug-Flow Furnace Model for Identification of Burner Thermal Behaviour

Zdeněk Jegla*

Institute of Process and Environmental Engineering, Faculty of Mechanical Engineering, Brno University of Technology, Technická 2, 616 69 Brno, Czech Republic
jegla@fme.vutbr.cz

High-performance tubular furnaces (fired heaters) of radiation-convection type, containing radiation chamber and convection section, are preferred for fundamental applications of process industry. The radiant chamber is dominant part of furnace since the most of heat is transferred to heated fluid there. Knowledge of heat flux distribution along the height of radiant chamber of tubular furnace is one of the crucial conditions of proper design of radiant tube coil system (especially when two-phase flow of heated fluid occurs inside radiant tubes). However, this information is not known in the initial design stage where the furnace design is performed according to relevant design standards and restrictions.
Paper presents actual results of development of so-called Modified Plug-Flow (MPF) furnace model for calculation and identification of real distribution of heat flux along height (length) of combustion chamber from burners. Obtained results show that MPF model is able to identify with sufficient accuracy a real burner thermal behaviour based on results of experimental measurement of real burners from burner testing facility.

1. Introduction

Fired heaters, boilers and furnaces are the major energy consumption components of industrial plants. Design and operating of these items is under continuous development, because optimum design of equipment leads to higher efficiency and, thus to lower operating cost, lower energy consumption, and lower amount of emissions being produced (Klemeš et al., 2010).
Present methods for optimum overall design and operating of fired heaters, boilers and furnaces and their integration into processes using low cost modelling techniques are well managed for different areas of industrial processes, such as, for example, fired heaters for refinery and petrochemical plant applications (Jegla, 2006), continuous steel-reheating furnaces (Tucker and Ward, 2012) or waste incineration boilers and furnaces (Jegla et al., 2010).
The radiant (combustion) chamber is a dominant part of fired heater, boiler or furnace since the most of heat is transferred to heated fluid there. Knowledge of heat flux distribution from burners along the height (length) of radiant chamber is one of the crucial conditions of proper design of radiant tube coil heat transfer system. It is very burning issue especially in the case of fired heaters in refinery applications (see Figure 1), when two-phase vapour-liquid flow of heated complex hydrocarbon mixture occurs inside radiant tubes. However, information about heat flux distribution is not known in the initial design stage where the fired heater design is performed according to relevant design standards and restrictions.
The level of agreement of applied design standard rules, especially of typical local distribution of heat flux along the radiant chamber height (length), and real thermal behaviour of installed burners in radiant chamber fundamentally influences operating behaviours of whole fired heater. When the level of agreement is not sufficient, serious operating troubles arise. Typically it is a separation of vapour and liquid phase of heated fluid and rapid deposition of coke inside tube coil resulting in overheating of tubes, rapid decline lifetime of exposed tubes, fast increasing of fluid pressure drop, and necessity of the plant shut down and decoking of fired heater or its revamp (Jegla et al., 2011).

Please cite this article as: Jegla Z., 2013, Development of modified plug-flow furnace model for identification of burner thermal behavior, Chemical Engineering Transactions, 35, 1195-1200 DOI:10.3303/CET1335199

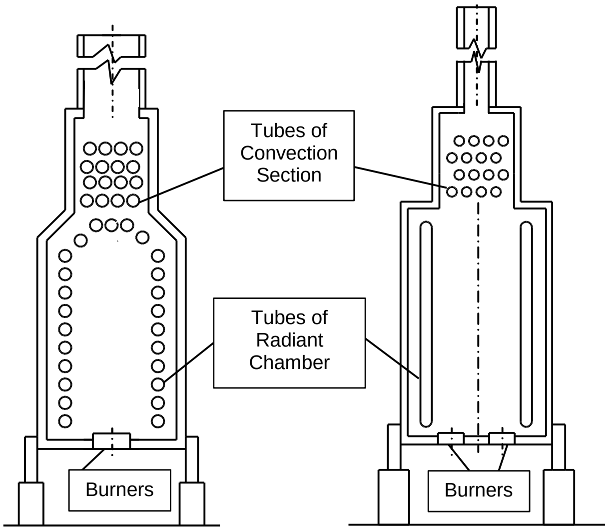

Figure 1: Typical refinery high performance radiation-convection types of fired heaters (cabin type – left, vertical cylindrical type - right)

Successfully solving of these problems consists in careful design of radiant tube coil system of fired heater. However, the reliable and verified information about the distribution of heat flux from the used burners along the height (length) of the radiation chamber of fired heater is necessary for careful radiant tube coil design.

2. The possibilities and difficulties of determination of heat flux distribution

Generally it is possible to determine a local heat flux along radiant chamber of fired heater by several approaches, their possibilities and obstacles will now be further discussed.
Experimental measurement on special testing facility - this option is only available to specialized scientific research institutes and manufacturers of burners. The burner is tested on a test combustion chamber equipped with detail and quality measurement and control system. The results of the test of burner allow to reliable identify, among other parameters, also the real distribution of heat flux low into the walls of the combustion chamber.
Prediction using computational fluid dynamics (CFD) software – using of detail numerical heat transfer approach for prediction of distribution of heat flux to walls in combustion chamber requires long time modelling experience with using CFD software. Results of CFD prediction have to be verified by experimental measurement on real burner. When the verified calculation model is used, results of CFD analysis give detail and real information about heat flux distribution.
Operational measurements - measurement on fired heater during its operation usually allows identifying the real local heat flux only at certain locations of radiation chamber. It does not so provide an overall picture of the distribution of heat flux along the chamber height (length). In addition, operating measurement requires special and expensive measuring equipment (most often thermography or heat flux meters) with trained operator (Jegla, 2012). Results of operating measurement are typically used as support information for heater operating analysis or for verifying of CFD models when information from the experimental test of burners is not available.
Prediction using low cost calculations – low cost calculation models are much more suitable for common fired heater design practice than CFD calculations, especially from time consumption reason. However, the information ability of available low cost models is much lower due to simplified mathematical formulations of these models. Potential of low cost models can be utilized for prediction of heat flux distribution after suitable modification of their formulation and completing with information from experimental testing of burners.
It is obvious from previous discussion that possible approaches to determine a distribution of heat flux along radiant chamber do not allow individually solve the problem and suitable combination of information from these approaches is necessary. One of the perspective and practically effective approach for identifying the heat flux distribution along height (length) of radiant chamber during design stage of fired heater we see in development of suitable form of low cost calculation model utilizing important information from experimental tests of real burners.

3. Experience with determination of heat flux distribution

Institute of process and environmental engineering (as part of R&D institution NETME Centre at Faculty Mechanical Engineering, Brno University of Technology) disposes besides experience with practical design and optimization of fired heaters also long time experience with determination of heat flux distribution especially in the areas of experimental measurement on special burner testing facility and prediction using CFD software. These activities will now be briefly introduced.

3.1 Experimental measurement on special burner testing facility

Main part of advanced burner experimental facility installed at Institute of process and environmental engineering creates a horizontal water-cooled combustion chamber with an internal diameter of 1 m and a length of 4 m, and with a maximum burner capacity of 1.8 MW. The cooling shell is split up to seven independent cylindrical sections with measured inlet and outlet water temperature and flow rates. This opens-up the possibility to evaluate heat fluxes to the walls along the flame. There are two inspection holes placed opposite each other in each section and two inspection holes at the end of the combustion chamber. The holes enable detailed observation of the flame and additional installation of measuring instrumentation. The experimental facility is equipped with a data acquisition system and advanced control system. The data acquisition system is designed to automatically collect all measured values including flow rates, temperatures and pressures. Facility serves for testing and development of burners for effective combustion of standard or renewable gas and liquid fuels. More details and parameters of burner testing facility can be found for example in Bělohradský et al. (2009) or Kermes and Bělohradský (2013).

3.2 Experience with prediction using CFD software

Long-time experience with using of CFD software FLUENT (Fluent, 2006) and its application to modelling, prediction and verification of heat flux distribution can be demonstrated ,for example on detail CFD analysis of behaviour of the low-NO_x gas burner of a diffusion-mixed type with forced combustion air supply and duty 745 kW. Detail description of tested burner can be found in Bělohradský et al. (2009). Geometrical model for CFD calculation including the air duct, the burner and the combustion chamber is shown on Figure 2.

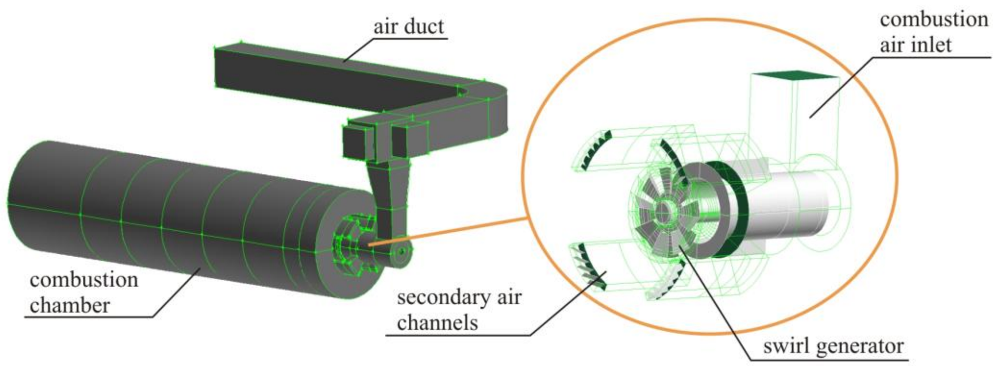

Figure 2: Model of combustion facility and detail of burner for CFD analysis, acc. to Bělohradský et al. (2009) - reprinted with permission

Detail computational analysis and prediction of turbulent, non-premixed swirling gas flame in an air-staged burner was performed in first stage of evaluation by Bělohradský et al. (2009) with average error of 15 % completed in second stage by Vondál and Hájek (2009) with average error of CFD model 10 %.

4. Development of Modified Plug Flow Furnace Model

Above mentioned experience with possibilities of determination of heat flux distribution results in development of low cost model suitable for common fired heater design practice for prediction of heat flux distribution utilizing information from experimental testing of burners.

4.1 Plug Flow Furnace Model

All commonly used low cost fired heater computational models can be utilizing principles of zone modelling technique. In the simplest form is model referred as well stirred (WS) furnace model, see, for example Hewitt et al. (1994). Furnace or radiant chamber of fired heater is in this model modelled in three zones, namely, the central hot gas (i.e. radiating flue gas) zone, the heat sink (i.e. zone representing tube coil system) and the zone of refractory walls.

Simplifying assumptions used in WS model are that the hot gases are perfectly mixed at a uniform temperature, T_g; the heat sink is grey and has a uniform temperature, T_1; and that the refractory surface is radiatively adiabatic (i.e. it reradiates all the heat that is receives). The WS model is adequate for predicting overall average heat transfer but provides no information on spatial variations, in particular those due to changes in heat release rate which occur as the gases flow through the furnace. This deficiency is significant when the length of the furnace is great in relation to its mean hydraulic diameter, for example, in case of (long or height) fired heaters, fire-tube boilers and tunnel kilns.

For the calculation of such firing equipment the plug flow (PF) *furnace model* can be applied. Instead of calculating the total rates of heat flow, as was done in the WS model, the furnace is considered as a (circular) channel (or duct) of length L divided into small segments of length dx, each of which is subject to local heat flux from the hot gases to the sink (q_1) and to local heat flux through the refractory wall to the outside of the furnace (q_r). The third factor required in the calculation is the local rate of heat release per unit length of tube, due to combustion. In the PF model it is assumed that the local volumetric heat release rate (h_f) is a known function of channel length direction x.

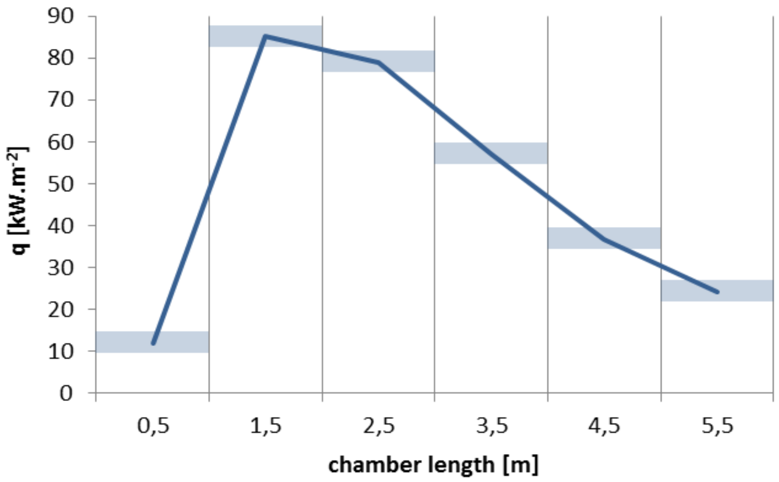

Figure 3: Heat flux profile along chamber length obtained with the PF model (data from Hewitt et al. (1994))

Knowledge of function h_f is, however, performable only in some cases, such, for example for continuous steel-reheating furnace (Tucker and Ward, 2012), where burners are placed in the certain length segments combusting fuel perpendicularly to the length axis of furnace. Thus according to burner duty it is possible to specify in these segments the parameter h_f.

For refinery fired heaters, such as vertical cylindrical type (see Figure 1, right part) the function h_f along height (length) of combustion chamber is unknown, because flame going in axis of chamber through several chamber length segments. Thus in such cases it is necessary to find function h_f along chamber height (length) from experimental test of given burner on testing facility and obtained results (knowledge of function h_f) use as input data for PF model (see chapter 3 above).

The same general assumptions are made in the PF model as in the WS case, namely, that all surfaces are grey and that refractory surfaces are in thermal equilibrium. The flow is assumed to be uniform; the gas temperature at any axial position is characterized by a single temperature, T_g; and the sink is characterized by a uniform temperature, T_1. However, in this case T_g varies with axial position dx. Finally, the assumption is also made that axial heat transfer by radiation can be neglected and that the beam length corresponding to a body of infinite length can be used in calculating gas emissivity (Hewitt et al., 1994).

Calculation procedure of PF model is based on the principle of iterative solution of heat balance and heat transfer in each length segment dx based on refinement of the estimated mean gas temperature T_g in the segment. The calculation goes sequentially through all length segments of solved combustion chamber. The resulting heat flux profile along length of combustion chamber is then obtained by connecting the calculated values of heat fluxes in centres of length segments, as shown in Figure 3 completing results of PF calculation example from Hewitt et al. (1994).

4.2 Development of Modified Plug Flow Model

Direct application of mathematical formulation of PF model to our experimental facility for burner testing (see chapter 3.1) shown that some general assumptions in the PF model not fully correspond with our burner testing facility situation and a PF model has to be modified. A significant difference is mainly that the testing facility inside combustion chamber contains no refractory walls, because all the inner surface of the test combustion chamber are absorbing surfaces (heat sink). From this reason estimated mean gas temperature T_g in the first segment, calculated according PF model gives too high value. The calculated final temperature of hot gases then it is too low and did not correspond to reality, which leads to an underestimation of the heat flux in the first segment, and conversely a significant increase in the size of the heat flux and temperatures in the following segments. After analysis of mathematical relations it was proposed following modification of calculation of mean gas temperature in the first segment:

$$T_{gM} = C_{Tg} \cdot T_g \tag{1}$$

where: T_{gM} is modified mean gas temperature in the first segment
T_g is mean gas temperature in the first segment calculated according PF model
C_{Tg} is correction factor for modification of mean gas temperature in the first segment, calculated as:

$$C_{Tg} = 1 - \frac{Q_{1w}}{Q_{Tw}} \tag{2}$$

where: Q_{1w} is heat absorbed by walls in the first segment of test combustion chamber
Q_{Tw} is total heat absorbed by walls of all segments of test combustion chamber

It is obvious from equations (1) and (2) that calculation procedure of such *modified plug flow (MPF) furnace model* is, more complicated, because evaluation of Q_{Tw} in eq. (2) is possible just after end of calculation sequence of all segments of combustion chamber. Thus, for first calculation of all segments the value of parameter C_{Tg} is set to 1 and is modified in next runs of calculation of all segments. I.e. MPF furnace model contains outer iterative loop for convergence of parameter C_{Tg}. The calculation of MPF model ends when substituting value of C_{Tg} in the calculation of the first segment coincides with the value of C_{Tg} obtained after the calculation of all segments.
Comparison of heat flux profile of low-NO_x gas burner with duty 745 kW (see Chapter 3.2) obtained from experimental measurement on burner testing facility with results of PF model and results of developed MPF model is shown on Figure 4.

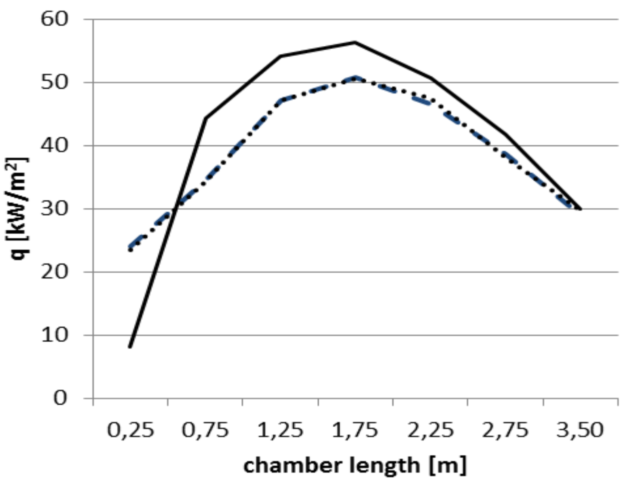

Figure 4: Comparison of heat flux profiles along length of test chamber for low-NO_x gas burner with duty 745 kW (Legend: —— PF model, – – – MPF model, ········ Measurement)

On the next Figure 5 is then presented heat flux profile of low-NO_x gas burner with duty 745 kW obtained from experimental measurement on burner testing facility in comparison with results of developed MPF model and with results of the best CFD model from Vondál and Hájek (2009).

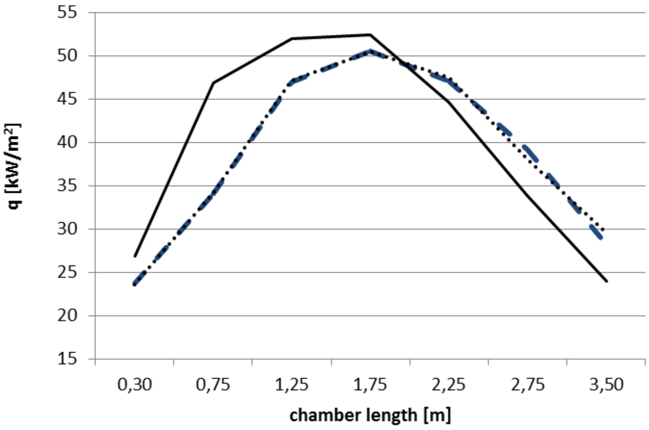

Figure 5: Comparison of measured and calculated (CFD, MPF) heat flux profiles along the test chamber for 745 kW low-NO_x gas burner (Legend: —— PF model, – – – MPF model, ········ Measurement)

5. Conclusions

Actual results of development of so-called Modified Plug-Flow (MPF) furnace model for easy calculation and identification of real distribution of heat flux along height (length) of combustion chamber are presented. Obtained initial results show that MPF model is able to identify with sufficient accuracy the actual burner thermal behaviour based on results of experimental measurements carried out at a burner testing facility. Validation of the developed MPF furnace model will be a part of the future work together with implementation of the model into improved fired heater design methodology.

Acknowledgement

Financial support of the Ministry of Education, Youth and Sports of the Czech Republic provided within the Operational Programme “Research and Development for Innovations”, project “NETME Centre – New Technologies for Mechanical Engineering” (reg. No. CZ.1.05/2.1.00/01.0002) is gratefully acknowledged.

References

Bělohradský P., Kermes V., Stehlík P., 2009, Computational modelling of swirling diffusion flame in air-staged burner, Book of abstracts - 4th European Combustion Meeting, 102-106, Verlag Process Engineering GmbH, Germany, ISBN 978-3-902655-06-6.

Fluent, 2006, FLUENT 6.3.26, User´s Guide, Fluent Inc., Lebanon, USA.

Hewitt G.F., Shires G.L., Bott T.R., 1994, Process Heat Transfer, Begell House, Inc., New York, NY, USA.

Jegla Z., 2006, The Conceptual Design of Radiant Chamber and Preliminary Optimization of a Process Tubular Furnace, Heat Transfer Eng. 27, 50-57.

Jegla Z., Bébar L., Pavlas M., Kropáč J., Stehlík P., 2010, Secondary Combustion Chamber with Inbuilt Heat Transfer Area - Thermal Model for Improved Waste-to-Energy Systems Modelling, Chemical Engineering Transactions, 21(2), 859 – 864, DOI: 10.3303/CET1021144.

Jegla Z., Kohoutek J., Stehlík P., 2011, Design and operating aspects influencing fouling inside radiant coils of fired heaters operated in crude oil distillation plants, Proceedings of International Conference on Heat Exchanger Fouling and Cleaning - 2011, June 05-11, Crete Island, Greece, www.heatexchanger-fouling.com/papers/papers2011/2_Jegla_F.pdf, accessed 21.3.2013

Jegla Z., 2012, Identification of operating load of tube coils inside fired heaters and boilers, Proceedings of the XIIIth International Conference Service Safety of Production Equipment in the Chemical Industry and Energetics, Bratislava, Slovakia, Lecture No.16, ISBN 978-80-227-3806-4.

Kermes V., Bělohradský P., 2013, Biodiesel (EN14213) heating oil substitution potential for petroleum based light heating oil in a 1 MW stationary combustion facility, Biomass and Bioenergy 49, 10-21.

Klemeš J.J., Varbanov P.S., Pierucci S., Huisingh D., 2010, Minimising emissions and energy wastage by improved industrial processes and integration of renewable energy, J. Clean. Prod. 18, 843-847.

Tucker R., Ward J., 2012, Identifying and quantifying energy savings on fired plant using low cost modelling techniques, Applied Energy 89, 127-132.

Vondál J., Hájek J., 2009, Experimental and numerical analysis of wall heat transfer in non-premixed gas combustor, Chemical Engineering Transactions, 18(1), 587 – 592, DOI: 10.3303/CET0918095.