CHEMICAL ENGINEERING TRANSACTIONS

VOL. 70, 2018

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

Guest Editors: Timothy G. Walmsley, Petar S. Varbanov, Rongxin Su, Jiří J. Klemeš

ISBN 978-88-95608-67-9; **ISSN** 2283-9216

DOI: 10.3303/CET1870246

Development of Heat and Fluid Flow Distribution Modelling System for Analysing Multiple-Distributed Designs of Process and Power Equipment

Zdeněk Jegla, Dominika Fialová*

Institute of Process Engineering, Faculty of Mechanical Engineering, Brno University of Technology, Technická 2, 61669 Brno, Czech Republic
dominika.fialova@vutbr.cz

More accurate modelling of heat and fluid flow distribution in apparatuses grows in importance due to ever-increasing demands on process and power heat transfer equipment such as heat exchangers, tubular furnaces, or steam boilers. The paper gives an overview of the currently available calculation methods and approaches to predicting and analysing flow behaviour and heat distribution in the most important types of process and power equipment. Properties, possibilities, and limitations of the individual calculation methods are discussed. Based on the analysis, the main findings from the development of the heat and fluid flow distribution modelling system for analysis of process and power equipment with multiple-distributed designs are presented. The proposed modelling conception is illustrated by employing an industrial case of an operated steam superheater with a specific multiple-distributed design. Additionally, future development of the intended fast yet accurate-enough modelling system for prediction of fluid flow and heat distribution is suggested.

1. Introduction

A considerable part of process and power heat transfer equipment operates on a short-term or long-term basis with uneven heat loading of its heat transfer surface together with non-uniform flow distribution of working fluids. These undesirable effects may arise from technological design and/or operating conditions of these apparatuses. Depending on the character of operating conditions, the most exposed parts of such unsatisfactorily designed equipment are subject to variously intensive manifestations of excessive operation. Consequently, the mechanical stress and deformation can lead to a total function failure of equipment.

As regards the distribution of heat and fluid flow in heat transfer equipment, a number of calculation methods, techniques, and approaches are available. Nevertheless, the issue of maldistribution is underestimated in a vast majority of cases. For example, the common thermal–hydraulic design of a tubular heat exchanger assumes that the process fluid is ideally distributed into the tubes, although the uniform flow distribution should be proven and investigated by available calculation methods. On one hand, a typical reason for avoiding the methods, which consider the maldistribution of heat and fluid flow in the thermal–hydraulic design of heat transfer equipment, is usually their long computational times or designers' uncertainty and distrust of the results obtained via these methods. On the other hand, there are still industrial heat exchangers with such complex specific design and construction that utilisation of common simplified methods for handling heat distribution together with distribution of process fluids is not possible.

The following text summarises main theoretical findings from the development of the heat and fluid flow distribution modelling system for analysis of heat and fluid flow distribution in process and power heat transfer equipment with multiple-distributed designs. The intention is to propose a method which would fill the space between the two "extreme" approaches – Computational Fluid Dynamics (CFD) and simplified analytic methods – to describe heat transfer equipment. The aim of the developed modelling system is to take advantages of both approaches and to yield the accurate-enough solution in a reasonable time frame. The findings are applied to an industrial case, which is a heat exchanger with the specific multiple-distributed design used to produce superheated steam.

Please cite this article as: Jegla Z., Fialova D., 2018, Development of heat and fluid flow distribution modelling system for analysing multiple-distributed designs of process and power equipment , Chemical Engineering Transactions, 70, 1471-1476 DOI:10.3303/CET1870246

2. Industrial heat exchanger with the specific multiple-channel system

Steam generators are one of the most frequent units in process and power industries. The industrial boiler partially discussed herein consists of multi-fuel firing (for liquid and gaseous fuels) and heat exchanger section. Superheated steam can be generated using at a wide range of pressures and temperatures depending on process or power applications. The questioned boiler has been employed in the chemical factory. Therefore, required technological parameters of the superheated steam are nominal pressure 3.8 MPa, nominal temperature approx. 375 °C and production of 16.7 kg/s (i.e. 60 tph). Due to challenging operating conditions it is impossible to avoid minor problems, but it is necessary to avoid more serious damages of the whole unit or some of its parts.

The most vulnerable part of the boilers are the superheaters (Jones, 2004). These heat exchangers are typically tailor-made systems with complex tubular design in an effort to compensate the generally low heat transfer coefficient of superheated steam, as well as flue gas, and to maximize heat transfer rate. The equipment is usually placed at the top of the furnace or as the first heat transfer surface in the second pass of the boiler, such as in a case of the superheater in question (see Figure 1).

Heat transfer equipment with complicated flow geometry tends to suffer from heat and fluid flow maldistribution. Non-uniform distribution of the hot stream (flue gas) in the shell side of the exchanger negatively affects heat loading of tubes, which are also badly influenced by uneven hot fluid flow across the tube bundle. Moreover, maldistribution of the tube-side fluid (superheated steam) can intensify the non-uniformity of heat transfer into the individual tubes. Once the uneven flow distribution occurs, there is a high risk of forming deposit layers which considerably impact thermal flow load of each tube in the bundle. Problematic distribution of process fluids can cause a decrease in the overall performance and, in the worst case, shorten the service life of the apparatus because of the tube failures. The respective superheater has undergone serious manifestations of heat and fluid flow maldistribution, namely the failure of tubes near the membrane wall, fouling on the tube side of the heat transfer surface, and the shutdown of a few channels.

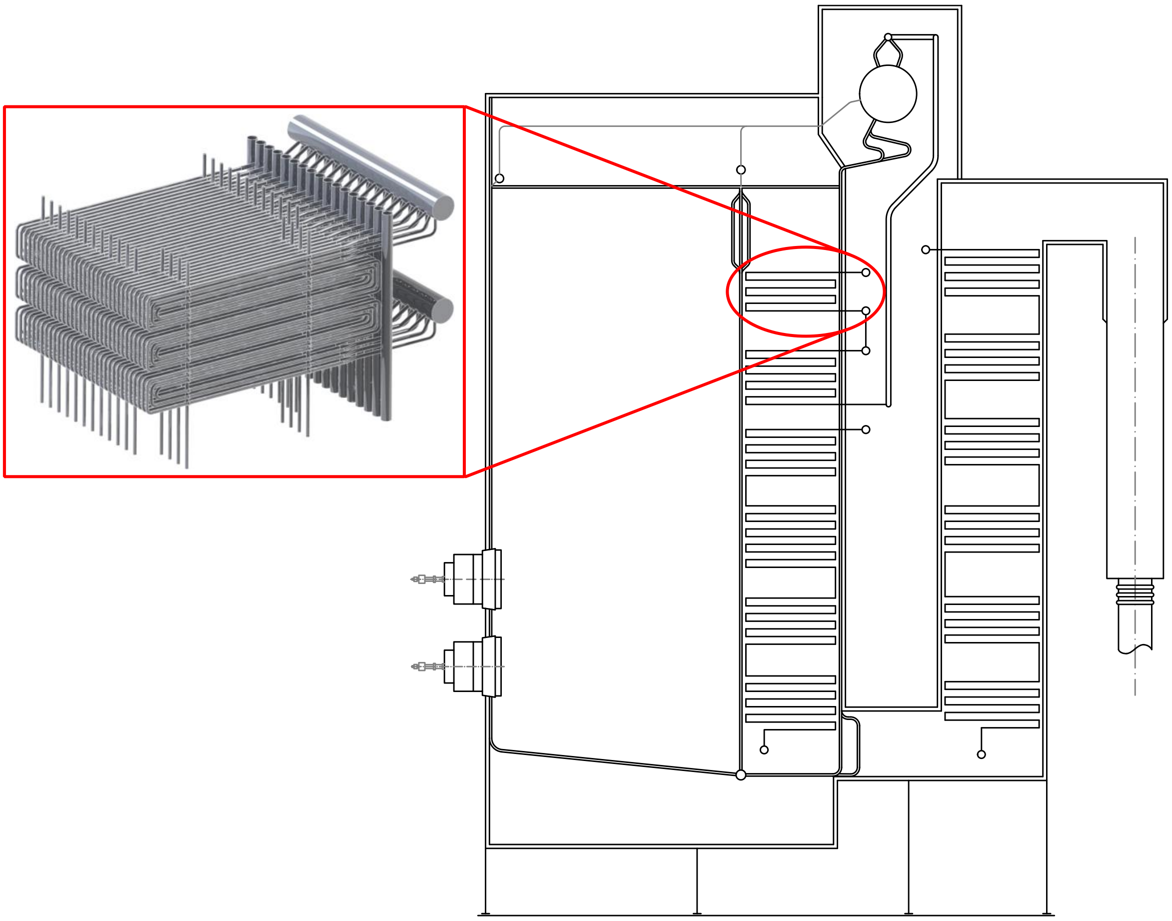


Figure 1: Schematic diagram of an industrial boiler and a model of the tubular steam superheater

As for construction, the specific superheater consists of splitting and collecting manifolds (so-called distributor and collector respectively) which are connected by 102 tubes in three inline rows. Some of the tubes are further split via “Y”-shaped elements, so in total, the tube bundle contains 198 tubes. The flow arrangement of the whole heat exchanger is one pass in a shell and six tube passes (see the part of the modelled steam superheater in Figure 2).

3. Common approaches to modelling of heat and fluid flow distribution

In general, there are three routes to model flow behaviour and heat distribution in heat transfer equipment. The first option is utilisation of analytical methods, which can predict heat and fluid flow distribution quickly via simplified means. On the other hand, large simplification limits the accuracy of these methods and their usage. Computational Fluid Dynamics (CFD) simulations enable one to investigate flow behaviour together with heat distribution issues in details anywhere within the modelled geometry. However, to yield precise data is extremely time-consuming, especially in case of complex 3D CFD models. A plausible compromise between the mentioned approaches is provided by hybrid methods which combines the lower computational requirements of simplified models and the high-quality data obtained by CFD analysis. So far, this challenging approach has been developed for only a few types of equipment with a rather simpler design of flow system.

Available simplified approaches for the further development of a new modelling system, which will comprehensively analyse heat transfer equipment with multiple-distributed design, usually focus on either fluid flow distribution, or on heat distribution. Under certain assumptions, flow distribution in heat transfer equipment can be spatially discretised in a very simple way, so a 1D-model description is more than sufficient. Limited flexibility, as well as the accuracy of (quasi-)1D approach, is compensated by low computational cost and very fast evaluation. As a result, these simplified methods are utilised mainly in the initial phase of a design process or in optimization tools. Nevertheless, more complex flow geometries have to be described by more demanding, yet still simplified, 2D or quasi-3D models (see e.g. Turek et al., 2015). In any case, it is crucial to verify simplifying assumptions by means of data from operating measurements (in the ideal case) or experimental data yielded by detailed CFD simulations.

Suitable 1D models able to evaluate flow distribution in the shortest time frames follow up the computational models presented by Bailey (1975), Bajura and Jones (1976), and Ngoma and Godard (2005). Bailey (1975) used a branch-by-branch approach to analyse a flow system without considering the change of fluid temperature. Bajura and Jones (1976) also utilised this approach for solving their isothermal model, though their description of additional momentum correction factors is not as advantageous as the coefficient of static regain defined by Bailey (1975). Due to implemented correction factors (the coefficient of static regains and the discharge coefficient), the simplified models are usually tailor-made to a specific design of apparatuses even with relatively complicated geometry including multiple tubes per loop, multiple-pass tube bundles, variable cross-section of the headers, etc. A quasi-1D mesh utilised by Ngoma and Godard (2005) fits particularly flow systems with significantly longer channels compared to their hydraulic diameters. Such spatial discretization is a satisfactory representation of equipment with rather simple geometry (see Figure 3a), but without additional correction factors this model is inapplicable to the steam superheater discussed herein (Figure 3b). However, this computational method demonstrates a possible approach to monitoring temperature profiles in manifolds and to modelling non-uniform heat flux into the tubes.

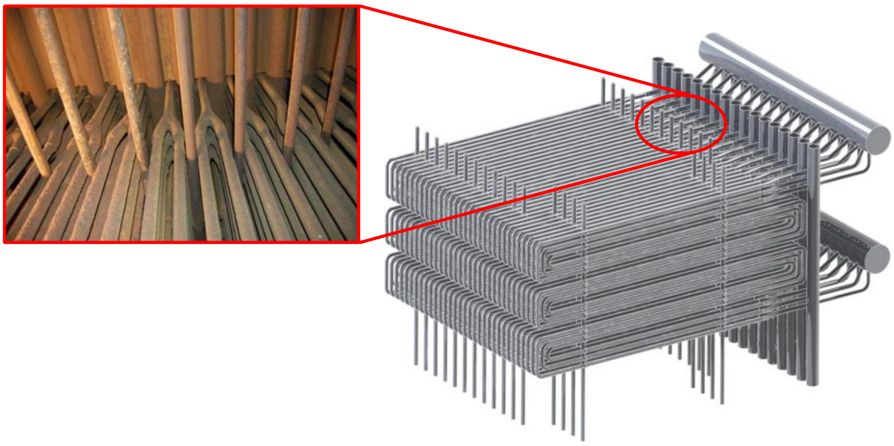
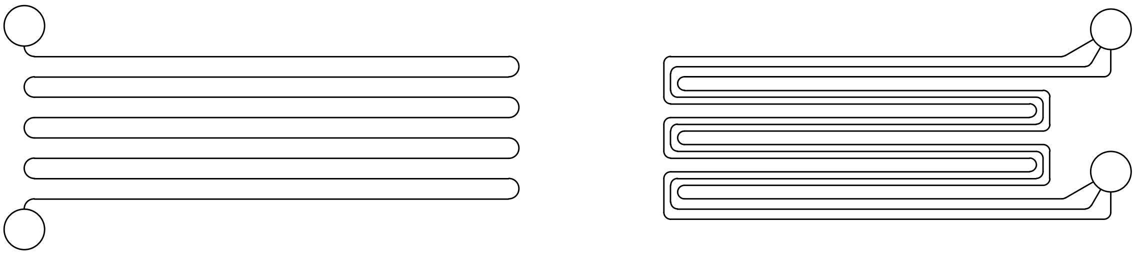


Figure 2: Modelled steam superheater and detailed view on “Y”-shaped tube elements



(a) (b)

Figure 3: (a) Channel system modelled by Ngoma and Godard (2005). (b) Channel system of the steam superheater discussed herein

As for heat distribution, acceptable calculation methods, which can obtain accurate-enough results in reasonable time frames, specialise in the specific types of heat exchangers. One of the well-established 2D calculation procedures is the Cell Method (Hewitt, 1998), which is suitable for shell-and-tube heat exchangers. The Cell Method makes it possible to predict temperature profiles of process fluids in heat exchangers with complex designs and constructions by dividing the heat exchanger into a number of simpler subexchangers (cells). This technique used on an example of an apparatus with segmental baffles and one longitudinal baffle is shown schematically in Figure 4.

Principles of another numerical analysis (Shah and Sekulić, 2003) are commonly used in the field of crossflow heat exchangers. A detailed investigation of heat transfer in the individual segments of discretised heat exchanger also takes into account the effect of fluid maldistribution. Limiting factors of this method are simplifying assumptions (such as single-pass heat exchanger configuration and unmixed–unmixed flow) which make it more difficult to utilise the method in the modelling of an equipment with a more complicated design. However, once all the important effects are incorporated into the calculation procedure, this approach will suit hybrid methods very well.

An example of such combination of CFD simulations and simplified methods is work presented by Starace et al. (2017) who considered a crossflow heat exchanger with enhanced heat transfer surface. The same principles as those presented in (Shah and Sekulić, 2003) were applied to create a 2D mesh, whereas various correction factors were provided by previous CFD simulations carried-out by Carluccio et al. (2005). It can be noticed from mentioned papers that development of a similar hybrid model using prediction functions is a real effort and, for the time being, the specific design and geometry of the steam superheater in question are too complex for this particular approach.

4. Outline of the modelling system and future work

The main purpose of the proposed modelling system is to describe comprehensively the fluid flow distribution in heat transfer equipment with complex design and construction, and in consequence, to utilise this information in calculating heat loading of heat exchanger channels. Although the modelling system aims to solve maldistribution of fluid flow on both sides of a heat exchanger, approaches to obtaining input data are different in the initial phase of the calculation.

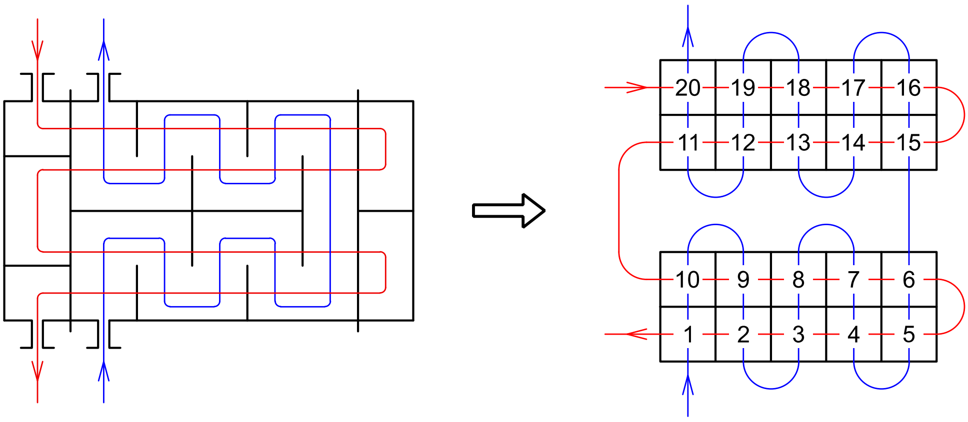


Figure 4: Cell Method applied on a shell-and-tube heat exchanger with longitudinal and segmental baffles

The maldistribution caused by variations in fluid properties (e. g. viscosity) due to temperature rise is not taken into account, because this phenomenon is typical rather for laminar flow (Mohammadi and Malayeri, 2013), and on the contrary, a vast majority of heat transfer equipment operates in turbulent flow regime. Once the temperature changes are not considered, the fast isothermal model based on the one presented by Bailey (1975) is perfectly sufficient for gaining input information about distribution of superheated steam. As for hot stream, temperature and velocity fields just above the discussed superheater, which were yielded via CFD simulations carried-out by Naď et al. (2017), are used as input data for a thorough investigation of the flue gas distribution in the shell side of the superheater.

Neither of available 2D heat distribution methods consider such complicated design and construction as those of the discussed apparatus. The created 2D mesh has to hold flow information across the tube bundle which contains six tube passes and three tubes per loop according to the classifying description by Rayaprolu (2012). For that reason, the crossflow method (Shah and Sekulić, 2003) is adjusted to the specific flow geometry by implementing the Cell-Method technique, i. e. to divide the equipment into a set of subexchangers. Each subexchanger then represents one tube pass – see a scheme in Figure 5, where the proposed modelling system is applied to the steam superheater geometry. Connections between the respective subexchangers (tube passes) are outlined as dotted lines.

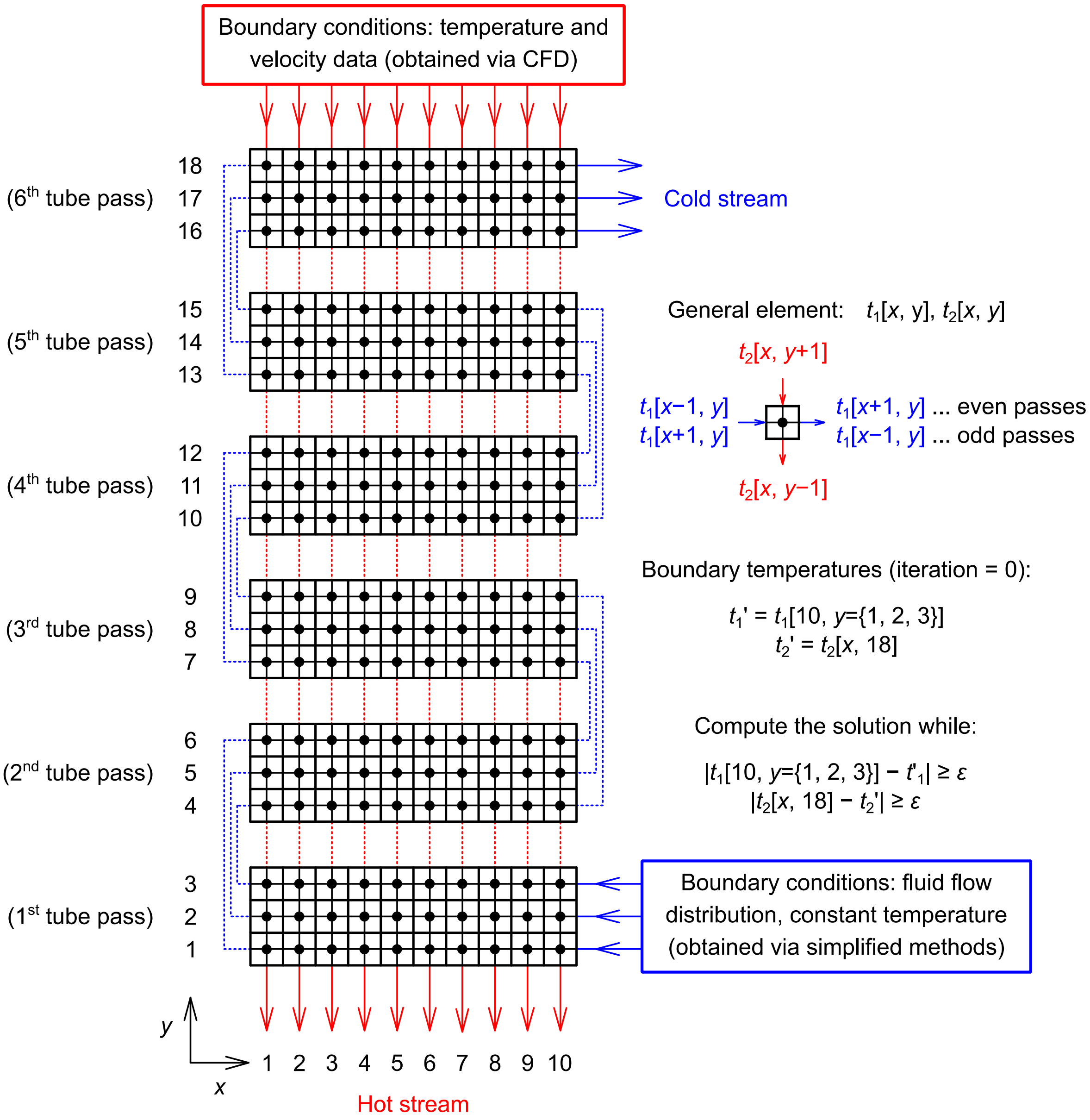


Figure 5: Proposed modelling system applied on the steam superheater and some basic calculation principles

The main principles of this calculation method (also in Figure 5) include: (i) boundary conditions (temperatures yielded via abovementioned approaches); (ii) temperatures relating to a general element; and (iii) a convergence criterion (the iteration procedure is finished once the absolute value of temperature difference in both inlet regions reaches the required accuracy, ε).

Apart from developing the combined 2D heat distribution method as described above, the emphasis of the future work will be put on validating the underlying assumptions. The first of them considers simplifications of the tube construction since the division of the tube-side flow via "Y"-shaped segments is neglected in the current model. Despite the fact that the respective superheater does not include any baffles in the shell side and mixing of hot fluid occurs when flue gas flows across the tube bundle, both available 2D heat distribution methods assume the hot fluid to be unmixed. Therefore, the influence of cold stream splitting, as well as the impact of mixed/unmixed hot fluid, on the accuracy of results has to be tested in upcoming activities. In this validation stage, data from experimental CFD simulations will be used, and additionally, the profitable feedback will be provided also by the comparison of the obtained results with the actual operating data.

5. Conclusions

A new modelling system for analysing heat and fluid flow distribution in heat transfer equipment with complex designs is introduced herein. On a case of an industrial steam superheater, a common part of process and power units, the main theoretical findings are presented. Available modelling approaches for predicting fluid flow behaviour, as well as heat distribution, are discussed using the specific heat exchanger, and consequently, the method described by Bailey (1975) has been chosen for further development of the proposed modelling system to evaluate tube-side flow distribution. The selected 2D crossflow method has been modified via the Cell-Method technique of subexchangers to model heat distribution in the complex flow geometry of the specific apparatus. Moreover, the new modelling system employs data from more general CFD simulations of the shell-side flow.

In future work, the integration of fluid flow and heat distribution will be followed-up by a validation of the resulting modelling system and also the accuracy issues arising from simplifying assumptions will have to be solved.

Acknowledgments

The authors gratefully acknowledge financial support within the project NETME CENTRE PLUS (LO1202) which was co-funded by the Ministry of Education, Youth and Sports within the support programme National Sustainability Programme I. The authors also sincerely thank for the financial support given by the Ministry of Education, Youth and Sports of Czech Republic under the standard specific research project FSI-S-17-4526.

References

Bailey B., 1975, Fluid flow in perforated pipes, Journal of Mechanical Engineering Science, 17, 6, 338–347.

Bajura R. Jones E., 1976, Flow distribution manifolds, Journal of Fluids Engineering, 98, 4, 654–666.

Carluccio E., Starace G., Ficarella A., Laforgia D., 2005, Numerical analysis of a cross-flow compact heat exchanger for vehicle applications, Applied Thermal Engineering, 25, 1995–2013.

Hewitt G. (Ed.), 1998, Heat exchanger design handbook, Begell House, New York, USA.

Jones D. R. H., 2004, Creep failures of overheated boiler, superheater and reformer tubes, Engineering Failure Analysis, 11, 873–893.

Mohammadi K. and Malayeri M. R., 2013, Parametric study of gross flow maldistribution in a single-pass shell and tube heat exchanger in turbulent regime, International Journal of Heat and Fluid Flow, 44, 14–27.

Naď M., Jegla Z., Létal T., Lošák P., Buzík, J., 2017, Thermal load estimation for superheater tube bundle design. in: Machine Modeling and Simulations MMS 2017, Žilina, p. 30, ISBN: 9788055413600.

Ngoma G. and Godard F., 2005, Flow distribution in an eight level channel system, Applied Thermal Engineering 25, 5–6, 831–849.

Rayaprolu K., 2012, Boilers: a practical reference, CRC Press, Boca Raton, FL, USA.

Shah R. K. and Sekulić D. P., 2003, Fundamentals of Heat Exchanger Design, John Wiley & Sons, Hoboken, NJ, USA.

Starace G., Fiorentino M., Longo M. P., Carluccio E., 2017, A hybrid method for the cross flow compact heat exchangers design, Applied Thermal Engineering, 111, 1129–1142.

Turek V., Fialová D., Jegla Z., Kilkovský B., 2015, Efficient 2D model of flow distribution in dense tube bundles, Chemical Engineering Transactions, 45, 1177–1182.