Open Access Journal

Journal of Power Technologies 93 (5) (2013) 295–302

journal homepage:papers.itc.pw.edu.pl

Numerical model of a cross-flow heat exchanger with non-uniform flow of media

Malgorzata Hanuszkiewicz-Drapala*, Tomasz Bury, Katarzyna Widziewicz

Institute of Thermal Technology
Silesian University of Technology, Gliwice, Poland

Abstract

This paper presents thermal-hydraulic analyses of finned cross flow heat exchangers working in media flow maldistribution conditions. The authors postulate a possibility of performing such analyses through the use of CFD models of recurrent segments of the heat exchangers. Media inflow to each recurrent segment may be individually defined and thus the flow maldistribution in the whole heat exchanger could be considered. The methodology of creating these models, running calculations and results of very initial experimental validation is presented in the paper.

Keywords: heat exchanger, flow maldistribution, thermodynamic analysis, numerical modeling, experimental validation

1. Introduction

Typical thermodynamic analysis of a heat exchanger is performed using some simplifying assumptions [1]. One of them is uniform flow of media through the device. However, this assumption is rarely fulfilled in reality and the influence of a non-uniform flow of media could be significant for heat exchanger performance.

The problem of media flow maldistribution in heat exchangers is far from new. It has been the subject of repeated investigations for many years. Results, especially taken from older works, are sometimes very unambiguous.

The first investigations referring to heat exchangers with media flow maldistribuiton were performed at the Institute of Thermal Technology of the Silesian University of Technology (ITT SUT) in respect of gaseous media and they were only computational in form [2]. Investigations of the gas-liquid type cross-flow heat exchanger have been conducted at the ITT SUT for a few years now with a view to evaluating the influence of a non-uniform gas inlet on exchanger functioning [3]. The range and form of air inflow non-uniformity were determined on a special test station—see Fig. 1. The configuration of the test station measuring system enables determination of the air velocity and temperature distribution at the heat exchanger inlet and outlet. This test station, in its original arrangement, only allowed for ”cold” experiments—meaning without presence of a hot medium. The influence of the measured non-uniformity was assessed through numerical simulations performed by the HEWES computer code—developed for thermal analyses of the considered heat exchanger. R. Piatek in his work [3] concludes

*Corresponding author
Email address:
malgorzata.hanuszkiewicz-drapala@polsl.pl
(Malgorzata Hanuszkiewicz-Drapala*)

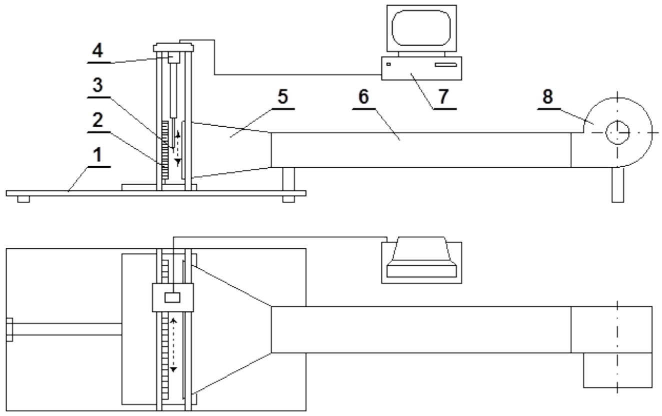

Figure 1: Test station—the air supply module (1—support plate, 2—heat exchanger, 3—thermoanemometric sensor, 4—measuring probe, 5—diffuser, 6—channel, 7—control computer, 8—fan)

that the maldistribution of the air inlet to the investigated car cooler may significantly impact the effectiveness of the heat exchanger.

Much of the research into the problem of non-uniform flow of media has been done only numerically. Authors of [4] simulated the plate fin heat exchanger using the finite elements method and discovered that the influence of non-uniformity of the liquid flow may be significant in some work regimes. A very significant drop in heat exchanger efficiency was also observed by the authors of [5]. Opposite results were obtained by the authors of [6] and [7]. Numerical simulations performed for a rotary heat exchanger in the first work and optimization procedure presented in the second one did not show significant dependence on flow non-uniformity of the agents. A common feature of works [5–7] is that the form and scope of the flow maldistribution were defined by the researchers.

Experimental analyses considering maldistributions of the agents flow through the heat exchangers and dealing with thermodynamic effects are rare. A Mueller in [8] concludes about the major significance of flow maldistributions for heat exchanger performance. Based on a study of gross flow maldistribution in an experimental electrical heater, the paper [9] presents the effect of flow non-uniformity on the performance of heat exchangers. The original fluid distribution is applied to heat exchangers (condensers, counterflow and cross-flow heat exchangers) and it is shown that gross flow maldistribution leads to a loss of effectiveness of about 7% for condensers and counterflow heat exchangers, and up to 25% for cross-flow exchangers. Similar effects observed by the authors of [10] indicate that non-uniformity affects the efficiency of heat exchangers to a large extent. Berryman and Russell studied flow maldistribution across tube bundles in air-cooled heat exchangers [11]. Their experimental results detected thermal degradation of up to 4%, which is much less than in previously cited works. The authors of [12] concluded about minor—up to 5%—effects of this phenomenon as well.

Very complex research was performed by teams from the Indian Institute of Technology—Madras and Lund University of Technology. These works concern plate-type heat exchangers. The numerical model of a one-pass plate heat exchanger was elaborated first for hydraulic analyses of flow maldistribution impact [13] and next it was arranged for multi-pass units [14]. An experimental investigation was also carried out to find the flow and pressure difference across the port to channel in plate heat exchangers [15]. More recently this research team did a thermal analysis too. The single-blow transient test technique based on the axial dispersion model was proposed for the determination of both the heat transfer coefficient and axial dispersion coefficient in plate heat exchangers. The experimental analysis presented in [16] deals with the effect of flow maldistribution on the transient temperature response for U-type plate heat exchangers. The experiments are carried out with uniform and non-uniform flow distributions for various flow rates and two different numbers of plates.

This literature review of the selected positions shows, as mentioned earlier, that the problem of non-uniform fluid inflow to the heat exchangers has been the subject of many computational and experimental investigations over the years, but the results obtained are unambiguous in terms of thermal performance. Many investigations are limited to hydraulic analysis only and they deal with liquid-liquid type heat exchangers. Most researchers are consistent in finding that non-uniformity of the flow significantly affects the hydraulic efficiency of heat exchangers. Thermal analyses refer first of all to heat exchanger ef-

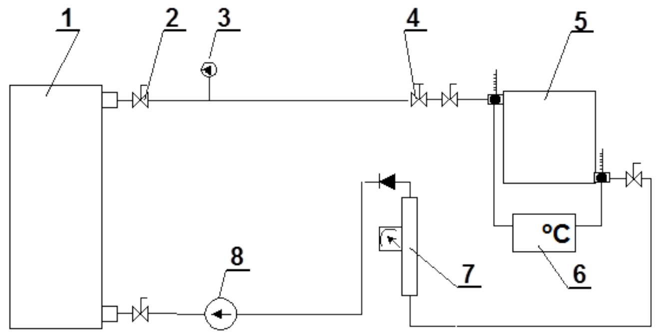

Figure 2: Test station—the hot water supply module (module (1—electric heater, 2—cut-out valve, 3—manometer, 4—control valve, 5—-heat exchanger, 6—temperature measuring system, 7—flow meter, 8—pump)

fectiveness, but they are not very numerous. There is a lack of complete investigations into finned cross-flow heat exchangers of the gas-liquid type with unequal inflow of the agents, especially of unequal inflow of gas. The detriment to heat exchanger efficiency could be significant (up to 18% according to results of numerical and experimental investigations in the ITT SUT). Therefore, in some particular analyses the non-uniformity of media flow should be taken into account.

2. Numerical model

2.1. Motivation

The first numerical model of a cross-flow finned heat exchanger considering media flow maldistribution was elaborated at ITT SUT some ten years ago [3]. This simplified model assumes division of a real device on recurrent elements and is based on the finite differences method. The model was next implemented into HEWES fast running computer code [3]. Experimental validation of this code becomes possible after some modifications of the test station, as mentioned earlier. The hot water supply module was installed—see Fig. 2—and the analyses done showed some important discrepancies between the experimental and numerical results [17].

One possible reason for this situation are the simplifications in geometry of the numerical model. This problem can be solved by applying CFD modeling methodology.

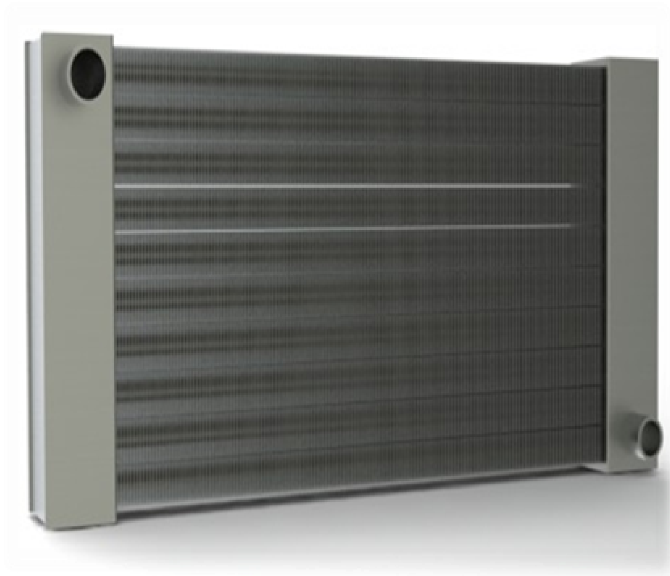

Figure 3: View of the analyzed heat exchanger

2.2. Basic assumptions

Building a CFD model of the whole finned heat exchanger is very time-consuming and the model would need very powerful computer to run it. Hence the authors postulate a possibility of using simplified 3D models of recurrent segments of the heat exchanger in order to consider the non-uniform flow of media and simulate the work of the whole device. The approach assuming division of the heat exchanger into recurrent elements is not new—a similar solution was used in [3]. D. Taler and his co-workers also used this approach [18, 19] and received very good agreement of the numerical and analytical solution, but media flow maldistribution was not considered. The difference is that a CFD model is able to reproduce the real geometry.

A very important issue when creating the numerical model is division of a real heat exchanger into recurrent segments. This process should be dependent on the heat exchanger construction. In the analyzed case the possibility of utilizing experimental data for model validation was also taken into account.

The analyzed heat exchanger takes the form of a fin and tube cross flow air water cooler and is presented in Fig. 3, similar design was considered in [20]. Its core is built of one row of elliptical pipes. The area of the air inflow is 490 mm wide and 280 mm high. Each of the 10 pipes is finned

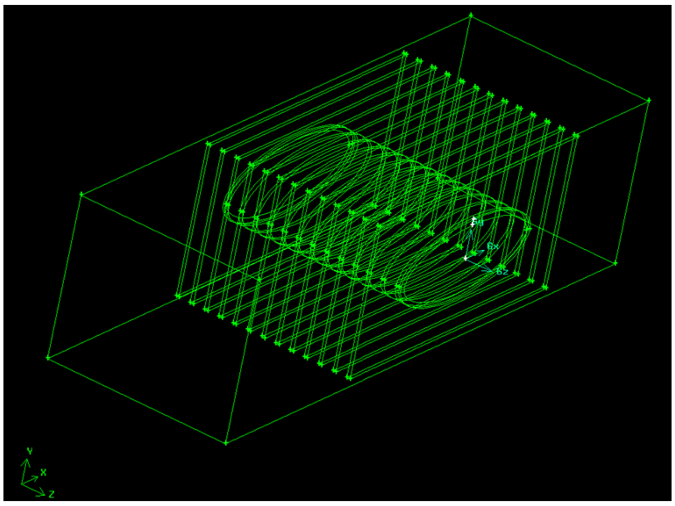

Figure 4: Geometry of the recurrent section of analyzed heat exchanger

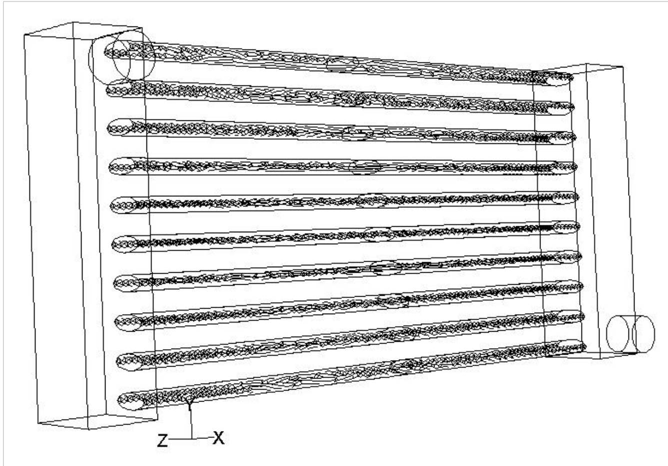

Figure 5: Geometry of the numerical model of water volume in the analyzed heat exchanger

with 175 flat rectangular fins spaced every 2.8 mm.

It was decided that a recurrent segment of the described heat exchanger should consist of a section of elliptical pipe with twelve fins. The geometry of the recurrent section is shown in Fig. 4.

The fin section in the model is surrounded by an air volume. The dimensions of the air volume are determined by the measuring system arrangement. The inlet plane is 40 mm ahead of the fin section, and the outlet plane is 19 mm behind it. These planes are located in the same position as the plane of thermoanemometric shifting. This makes it possible to use experimental data as the boundary conditions and for validation too. The water volume inside the pipe section is also simulated. The numerical mesh of this recurrent segment is unstructural and numbers nearly 920,000 T-Grid cells.

A set of basic governing equations was added to this numerical mesh to complete the numerical model. This set includes:

- Energy balance equation
- Continuity equation (mass balance)
- Momentum equation
- Equations of state (for determining thermophysical properties of media)
- Turbulence model

The velocity inlet boundary condition was assigned to the air inlet plane. This makes it possible to use experimental data and take into account air inflow maldistribution. The mass flow inlet condition was set for water inflow. Both air and water outlets were set as the ”outflow” boundary conditions. From previous experience (works [3] and [18]) the side walls of the model were treated as symmetry planes. One of the basic assumptions here is that the recurrent section has negligible flow interactions with neighboring sections and hence may be treated as a separate flowing channel.

Initial simulations performed for the described numerical model informed the choice of turbulence model. The standard $k - \epsilon$ model was checked as well as the Reynolds Stress Model. The former was chosen as it gave physically correct and satisfactory results in a reasonable time.

2.3. *Water flow maldistribution*

Non-uniform water flow is another problem in the heat exchangers under consideration. Unfortunately, the form and scope of this non-uniformity cannot be determined experimentally in the present arrangement of the test station. Therefore an additional CFD model of the water volume for the heat exchanger was elaborated. The geometry of that model is shown in Fig. 5. It consists of inlet and outlet water headers and ten elliptical pipes. The model is used mostly for hydraulic simulations and the aim is to determine the mass flow rates of water in each pipe.

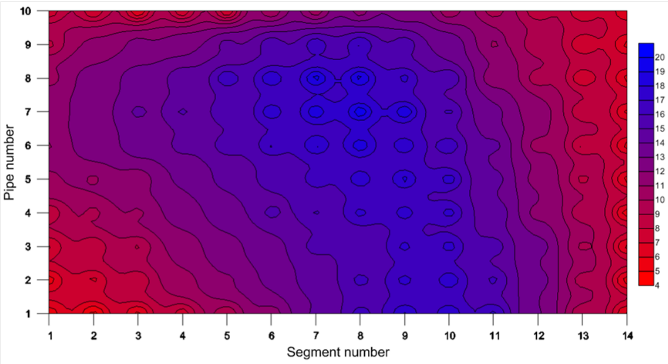

Figure 6: The air inlet velocity distribution for the maximum air flow rate, m/s

The numerical mesh consists of about 980,000 cells. Testing computations realized in the first step showed that both the $k-\varepsilon$ and RSM turbulence models seem to be appropriate for this case. As for the recurrent segment model, as here the $k-\varepsilon$ model was chosen.

3. Results of simulations

3.1. Analyzed cases and boundary conditions

Two experiments were simulated using the above described numerical models. The experiments were realized at the maximum inlet temperature of water set at 90°C and maximum water flow rate—about 26 liters per minute. The first measurement was taken at the maximum air flow rate and the second one at the minimum air flow rate.

The most important experimental result is the air inlet velocity distribution. It was measured by the thermoanemometric sensor in 180 points located on the inlet plane. The dimensions of the sensor shifting area are slightly smaller than the dimensions of the air inflow area. This is due to a risk of damage to the sensor caused by its collision with inlet diffuser walls. Thus, the experimental results were extrapolated in order to obtain the air velocity field for the whole area. This velocity distribution stands as the velocity inlet boundary condition attained to the numerical model. Fig. 6 presents the air inlet velocity field for the first experiment (with maximum air flow rate). This velocity field is presented for 140 recurrent segments creating the whole analyzed heat exchanger, so the numbers on the vertical axis are the numbers of pipes and on the horizontal axis the labels mean the number of segments on a pipe.

Table 1: Results of measurements

Parameter	Measurement 1	Measurement 2
$\dot{V}_a$, m^3/h	6100	3650
$\dot{V}_w$, dm^3/min	26.0	26.2
$t_{a\,in}$, °C	26.0	26.3
$t_{a\,out}$, °C	34.3	37.6
$t_{w\,in}$, °C	88.1	89.8
$t_{w\,out}$, °C	79.4	82.3
$\dot{Q}_a$, kW	16.83	13.93
$\dot{Q}_w$, kW	15.46	13.33

It can be clearly observed that the air inflow is non-uniform, and the maximum recorded velocity is almost four times higher than the minimum one.

Important data in the form of the total heat transfer rates for the heat exchanger were given by the experiments. This parameter could be calculated as the air enthalpy increase:

$$\dot{Q}_a = \dot{V}_a \rho_a c_p \left(t_{a\,out} - t_{a\,in}\right) \tag{1}$$

where: $\dot{V}_a$—volumetric air flow rate, ρ_a—density of the air, c_p—specific heat capacity at constant pressure, $t_{a\,in}$, $t_{a\,out}$—air temperature at the inlet and outlet respectively.

These temperatures are calculated as the area weighted average values for the inlet and outlet planes. However, there is a problem: the volumetric air flow rate is determined at the end of rectangular flow channel and there are some leaks of air before the heat exchanger (mostly through the measuring probe shifting hole). Therefore it is better to compute the total heat transfer rate as the water enthalpy drop:

$$\dot{Q}_w = \dot{V}_w \rho_w c_{pw} \left(t_{w\,in} - t_{w\,out}\right) \tag{2}$$

where: $\dot{V}_w$—volumetric water flow rate, ρ_w—density of water, c_{pw}—specific heat capacity at constant pressure, $t_{w\,in}$,$t_{w\,out}$—water inlet and outlet temperatures.

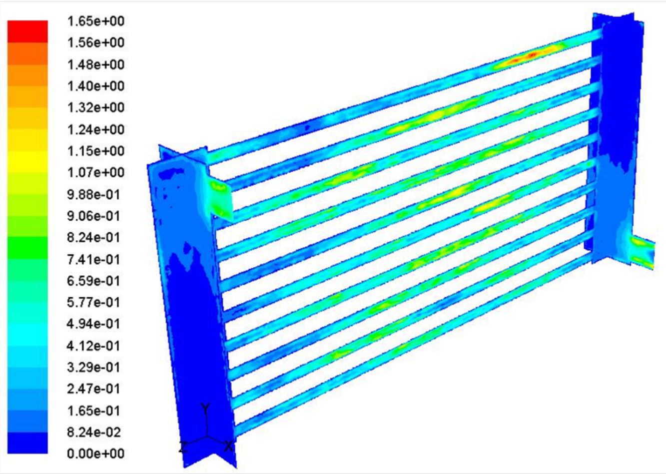

Figure 7: Water velocity distribution in the analyzed heat exchanger

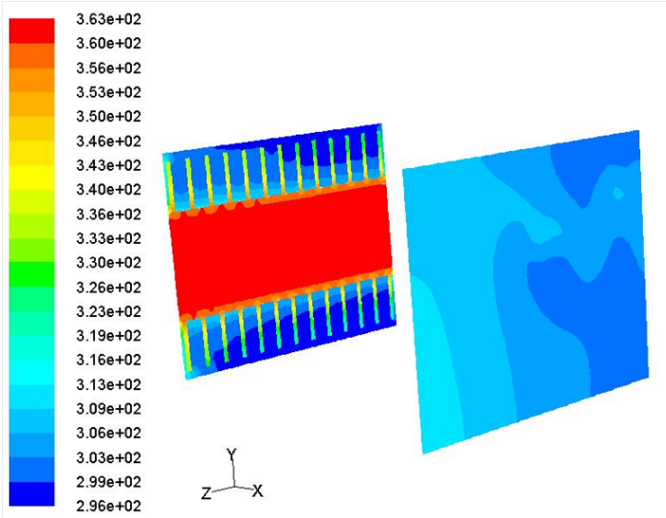

Figure 8: Air and water temperature distributions in the middle of the fins section and the air outlet temperature, K

Table 2: Water distribution among the pipes (in kg/s)

Pipe No	Measurement 1	Measurement 2
1	0.0459	0.0462
2	0.0478	0.0481
3	0.0468	0.0471
4	0.0449	0.0452
5	0.0431	0.0434
6	0.0412	0.0415
7	0.0393	0.0396
8	0.034	0.0386
9	0.0365	0.0368
10	0.0375	0.0377
Total	0.4214	0.4242

The above mentioned parameters were calculated for both measuring series and the results are set out in Table 1.

Water flow distribution among the pipes was determined by using the hydraulic model of the heat exchanger under consideration. Fig. 7 presents water velocity fields in three planes crossing the model in symmetry axes of the elliptical pipes and the inlet and outlet nozzles. It can be seen that water distribution is non-uniform and slightly more water flows through the upper pipes, as expected due to the location of the inlet nozzle.

Mass flow rates of water flowing in each pipe were computed according to the numerical results and are presented in Table 2. The relative difference between the maximum and minimum mass flow rates in the pipe exceeded 30%.

The mass flow rates of water given in Table 2 were put into the numerical model of the recurrent segment as the boundary conditions for water.

3.2. *Selected results of simulations*

The main parameter of interest is the total heat transfer rate. It was assumed that if the model is able to predict heat exchanger performance in non-uniform media flow conditions, it would also be able to predict the performance of the same heat exchanger with uniform flow of agents. This will open up the possibility of assessing the deterioration in efficiency. The total heat transfer rate is the sum of heat transfer rates for individual recurrent segments. This parameter was calculated for both analyzed cases and is presented in Table 3. One can perform a very preliminary validation of the model by comparing the experimental and numerical heat transfer rates.

The CFD analysis produces many interesting results, such as the temperature distributions (see Fig. 8), but in order to extend the validation process the following parameters were also compared:

- Average air outlet temperature
- Distribution of the air outlet temperature
- Average water outlet temperature

Table 3: Results of simulations

Parameter	Case 1	Case 2
$\dot{Q}_{w\,ex}$, kW	15.46	13.33
$\dot{Q}_{w\,num}$, kW	14.86	12.49
δ_Q, %	3.9	6.3
$t_{a\,out\,ex}$, °C	34.3	37.6
$t_{a\,out\,num}$, °C	32.9	35.9
$\delta_{t_{a\,out}}$, %	4.1	4.5
$t_{w\,out\,ex}$, °C	79.4	82.3
$t_{w\,out\,num}$, °C	76.2	78.9
$\delta_{t_{w\,out}}$, %	4.0	4.1

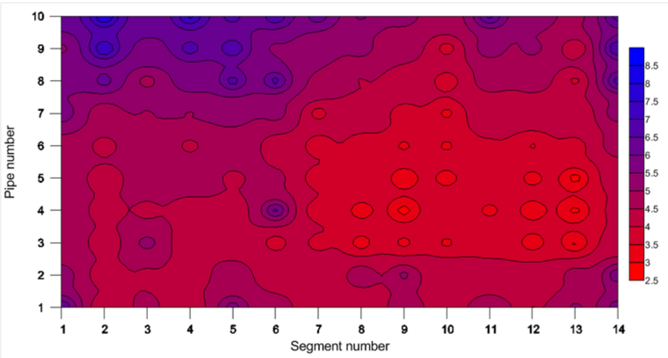

Figure 9: Distribution of the relative differences for the air outlet temperature—case 2

The average air outlet temperature was computed as the arithmetic average for all 140 recurrent segments. The average outlet temperature for water was calculated as the mass flow rate average value.

Relative differences were determined for all compared parameters using the following general formula:

$$\delta_X = \frac{X_{ex} - X_{num}}{X_{ex}} \tag{3}$$

where X_{ex} denotes an experimental value of compared parameter and X_{num} a numerical value. The relative differences are shown in Table 3. Case 1 in Table 3 refers to measurement 1 and case 2 refers to measurement 2. Analysis of the results shows that the numerical values are underestimated. The relative differences of the total heat transfer rates are about 5% on average and may be viewed as satisfactory compared with those previously obtained from the HEWES code (6...11%, see [17]).

The relative differences map for the air outflow temperature is shown in Fig. 9. Larger differences appeared in areas where the air inflow rates are lower and this suggests that turbulent flow may not be fully developed.

4. Conslusions

The work presents an alternative approach to thermal-hydraulic analyses of non-uniform flow of media in finned cross flow heat exchangers. While the approach may seem a little innovative, building a numerical model of the whole finned cross flow heat exchanger under consideration with a fine enough numerical mesh would mean creating about 130 million cells and is therefore unmanageable on a typical desktop computer. Dividing the heat exchanger into recurrent segments and running sequence computations could provide a reasonable solution.

The results of the initial numerical analyses and validation process confirms the general assumptions of the model. Further validation of the models is required as are some improvements in the models (finer numerical mesh).

Acknowledgements

The experimental part of this work has been realized within the framework of project N512 458836 in 2009...2012. The numerical analyses were performed with financial support from the state funds of the Department of Energy and Environmental Engineering of the Silesian University of Technology.

References

[1] A. Baranowski, J. Ferdyn-Grygierek, Impact of air exchange on the heat consumption in dwelling houses and public buildings, Rynek Energii 107 (4) (2013) 85–89.

[2] M. Hanuszkiewicz-Drapała, Thermodynamic analysis of a cross-flow heat exchanger with non-uniform flow of agents, Ph.D. thesis, Silesian University of Technology, Institute of Thermal Technology, Gliwice, Poland, (1996).

[3] R. Piątek, Thermal analysis of plate fin and tube heat exchanger with unequal inlet of media, Ph.D. thesis, Sielsian University of Technology, Institute of Thermal Technology, Gliwice, Poland (2003).

[4] K. Ranganayakulu, Ch.and Seetharamu, K. Sreevatsan, The effects of inlet fluid flow nonuniformity on thermal performance and pressure drops in crossflow plate-fin

compact heat exchangers, International Journal of Heat and Mass Transfer 40 (1) (1996) 27–38.

[5] M. Andrecovich, R. Clarke, Simple modeling of flow maldistribution in plate-fin exchangers, in: Proceedings of the 21st IIR International Congress of Refrigeration, Paper ICR0639, Washington DC, USA, 2003.

[6] S. Nair, S. Verma, S. Dhingra, Rotary heat exchanger performance with axial heat dispersion, International Journal of Heat and Mass TransferInternational Journal of Heat and Mass Transfer 41 (18) (1998) 2857–2864.

[7] K. Lee, S. Oh, Optimal shape of the multi-passage branching system in a single-phase parallel-flow heat exchanger, International Journal of Refrigeration 27 (1) (2004) 82–88.

[8] A. Mueller, Effects of some types of maldistribution on the performance of heat exchanger, Heat Transfer Engineering 8 (2) (1987) 75–86.

[9] S. Lalot, P. Florent, S. Lang, A. Bergles, Flow maldistribution in heat exchangers, Applied Thermal Engineering 19 (8) (1999) 847–863.

[10] X. Luo, W. Roetzel, U. Lüdersen, The single-blow transient technique considering longtitudinal core conduction and fluid dispersion, International Journal of Heat and Mass Transfer 44 (1) (2001) 121–129.

[11] R. Berryman, C. Russell, The effect of maldistribution of air flow on aircooled heat exchanger performance, Maldistribution of flow and its effect on heat exchanger performance, J.B. Kitto, J.M. Robertson ASME Htd 75 (1987) 19–23.

[12] C. Meyer, D. Kröger, Plenum chamber flow losses in forced draught air-cooled heat exchangers, Applied Thermal Engineering 18 (9-10) (1998) 875–893.

[13] N. Srihari, B. Rao, B. Sunden, S. K. Das, Transient response of plate heat exchangers considering effect of flow maldistribution, International Journal of Heat and Mass Transfer 48 (15) (2005) 3231–3243.

[14] N. Srihari, S. K. Das, Transient response of multi-pass plate heat exchangers considering the effect of flow maldistribution, Chemical Engineering and Processing: Process Intensification 47 (4) (2008) 695–707.

[15] B. Rao, B. Sunden, S. K. Das, An experimental investigation of the port flow maldistribution in small and large plate package heat exchangers, Applied Thermal Engineering 26 (16) (2006) 1919–1926.

[16] K. Shaji, S. Das, The effect of flow maldistribution on the evaluation of axial dispersion and thermal performance during the single-blow testing of plate heat exchangers, International Journal of Heat and Mass Transfer 53 (7-8) (2010) 1591–1602.

[17] T. Bury, J. Składzień, R. Piątek, Validation and sensitivity analysis of the mathematical model of a cross-flow heat exchanger with non-uniform flow of agents, Systems - Journal of Transdisciplinary Systems Science 13 (1) (2008) 19–27.

[18] D. Taler, A. Cebula, Numerical modeling of the heat transfer in the lamel type heat exchangers, in: Proceedings of the 12th Symposium on Heat and Mass Transfer, Vol. 2, Kraków, Poland, 2004, pp. 827–836.

[19] D. Taler, A. Sury, Mathematical models of pipe type cross-flow heat exchangers and application for regulation of media outlet temperature, Cracow University of Technology, MonographCracow University of Technology, Monograph, Mechanics series (437).

[20] K. Futyma, M. Wołowicz, J. Milewski, W. Bujalski, J. Lewandowski, Utilization of flue gas low temperature heat from the power unit, Rynek Energii 96 (2011) 74–79.