Experimental and numerical analysis of the impact of a liquid flow rate on the operational performance of a cross-flow tube-and-fin heat exchanger

TOMASZ BURY[*]
MAŁGORZATA HANUSZKIEWICZ-DRAPAŁA

Silesian University of Technology, Department of Thermal Technology, Konarskiego 22, 44-100 Gliwice, Poland

Abstract The paper is dedicated to an issue of the influence of a non-uniform flow of mediums in a cross-flow water-air heat exchanger, the core of which is a bundle of elliptical finned tubes. The main purpose of the work is to determine the impact of non-uniform water inflow for various mass flow rates on the thermal efficiency of the heat exchanger. Multivariate analyses were carried out for various temperatures of water, and for measured non-uniform air distribution at the heat exchanger input. Two variants of water distribution were considered: non-uniform water distribution assumed considering a non-uniform air inflow and water distribution resulting from hydraulic resistances calculated for different locations of water inlet and outlet nozzles. Simulation results were compared with the experimental outcomes obtained in cases of the non-uniform natural inflow of both mediums and to the computation results for a case of the uniform media inflow. The results obtained in this work confirm the significant deterioration of the thermal efficiency of heat exchangers caused by a non-uniform media inflow (by as much as 18.5% compared to the case of a uniform media inflow) which is compliant with other numerous works. The control of the water flow through the individual heat exchanger tubes enables the improvement of thermal efficiency by 4.5% to 18.6% compared to the device with uncontrolled inflow of working fluids, which for some of the analyzed cases is even better than a completely uniform inflow of heat carriers.

[*]Corresponding Author. Email: tomasz.bury@polsl.pl

Nomenclature

$C_{1\varepsilon}$, $C_{2\varepsilon}$, $C_{3\varepsilon}$	–	constants
c	–	specific heat capacity, $\mathrm{J/(kg\,K)}$
G_k	–	turbulence kinetic energy generation due to the mean velocity gradients, $\mathrm{kg/(m\,s^3)}$
G_b	–	turbulence kinetic energy generation due to buoyancy, $\mathrm{kg/(m\,s^3)}$
k	–	turbulent kinetic energy, $\mathrm{J/kg}$
$\dot{Q}$	–	heat flux, W
S_k, S_ε	–	source terms, $\mathrm{kg/(m\,s^3)}$
t	–	temperature, °C
u	–	velocity, m/s
$\dot{V}$	–	volumetric flow rate, $\mathrm{m^3/s}$

Greek symbols

ε	–	dissipation rate, $\mathrm{J/(kg\,s)}$
μ	–	dynamic viscosity, Pa·s
ρ	–	density, $\mathrm{m^3/kg}$
σ_k, σ_ε	–	turbulent Prandtl numbers

Subscripts

a	–	air
in	–	at the heat exchanger inlet
out	–	at the heat exchanger outlet
t	–	turbulent
w	–	water

1   Introduction

Experimental and computational studies of heat exchangers show that the flow of working mediums through tube and fin cross-flow heat exchangers can be non-uniform. A non-uniformity (maldistribution) may be, among others, a result of a heat exchanger design, methods of forcing flows of heat carriers, or the shape of inlet channels of mediums. This problem is not new and has been the subject of analyses for many years, but past studies [1,2] have focused on assessing the importance of the form of a maldistributed flow of mediums on the thermal efficiency of heat exchangers. Analyzing available publications, two important aspects have to be highlighted, which actually state a motivation for this work. First, a non-uniform inflow of

working fluids may significantly deteriorate the performance of a heat exchanger compared to a uniform inflow of media. Second, it is possible to improve the performance of a heat exchanger with flow maldistribution by controlling the inflow of mediums.

Minichannel heat exchangers are the subjects of many more recent works. A hydraulic analysis of a minichannel device is presented in [3]. The authors built a computational fluid dynamics model of a minichannel heat sink and carried out a set of simulations to determine the effects of inlet/outlet nozzles' location and flow rate on flow maldistribution. Numerical and experimental investigations on microchannel heat exchangers were realized by Joseph with his co-workers [4]. They found out that the flow maldistribution strongly influenced the thermal performance of the considered heat exchanger. They also indicated the possibility of enhancing the heat exchanger efficiency by changing the flow rate of a working fluid. The problem of medium flow non-uniformity and its meaning to heat exchangers' operational performance was also investigated in [5], in which authors proposed a fast method for evaluating the effect of this phenomenon on the thermal performance of heat exchangers. A proposed method is based on their study findings, which were transformed into a plot. The plot allows the evaluation of flow maldistribution impact on thermal performance for different velocity deviation patterns.

The considered problem of medium flow maldistribution is significant and T'joen *et al.* [6] proposed a numerical tool supporting the design process of heat exchangers taking into account the non-uniform flows of working fluids. The need to take this phenomenon into account for the design of more efficient devices was also stated in the publication of Tereda *et al.* [7]. The research results published so far concern various types of heat exchangers, but the common conclusion is that the non-uniform flow of working fluids can significantly deteriorate the efficiency of a heat exchanger in certain operating modes. Three types of non-uniformities were investigated by De Schampheleire *et al.* [8] with regard to the air-water plate heat exchanger, who found up to 25% increase in heat transfer resistance compared to the uniform case. On the other hand, studies of a cross-current heat exchanger with supercritical pressure carbon dioxide as the working medium carried out under the work of Guo *et al.* [9] showed that maldistribution of the medium inflow and its temperature at the inflow can both worsen and improve the conditions of heat transport in the device. In addition, the authors stated that these positive or negative effects occurring on both sides of the exchanger's partition can be cumulative.

The issue of the non-uniform inflow of mediums to heat exchangers was also investigated by the authors of the present paper. The experimental and computational analyses concerned cross-current tube-and-fin liquid-gas heat exchangers [10,11]. The non-uniformity applies to both fluids (gas and liquid), but due to technical limitations, the range and form of non-uniformity were determined by the gas side measurement only [10,11]. The results of analyses are consistent with the results of other researchers and indicate that the non-uniformity of the mediums inflow may deteriorate the efficiency of the device by up to 20% compared to the heat exchanger operating at uniform inflow of working agents.

The thermal efficiency of any heat exchanger depends on many parameters, such as device dimensions, materials, and operational characteristics. Extending the heat exchange surface is the simplest way to increase the efficiency of the heat exchanger. On the other hand, the size of the device is usually limited because it has to fit in with the rest of the installation. Therefore, other methods of improving the efficiency of such devices without increasing the heat transfer surface are considered. This can be achieved by using active and passive methods, or a combination of both [12]. Passive methods are of particular interest due to relatively low costs and no need for additional energy supply. Such methods often involve modifying the geometry of the heat exchanger or using inserts that increase turbulence in the flow of working fluids. It should be noted, however, that such solutions increase the pressure drop in the flow of factors, which usually also increases the driving power demand of the devices forcing the flow. Coetzee *et al.* [13] performed a series of measurements on a tube-in-tube heat exchanger using a set of spiral inserts. They achieved an increase in the heat exchanger efficiency of 206%, but at the same time, the pressure drop increased by 203%. It should be noted that research on the effectiveness of the passive methods is most often conducted separately from the issue of non-uniform inflow of mediums to heat exchangers.

Bury and Składzień [14] also examined the effectiveness of the use of inserts increasing the degree of turbulence of the gas at the heat exchanger inflow. However, the tests carried out for the cross-current, finned air-water cooler gave negative or inconclusive results. In the subsequent stages of the research, it was assumed that the non-uniform but controlled inflow of media may have a positive effect on the heat exchanger's operation. A similar approach, using the shaping of incoming media streams, was also analyzed by Hajabdollahi *et al.* [15]. They optimized the design of the heat exchanger in terms of the structure of the fins for different inlet

profiles of the two mediums. It was found that it was possible to obtain a thermoeconomic advantage over the exchanger with the uniform inflow of heat carriers. The authors of the present paper conducted analyses of the possibility of increasing the thermal efficiency of a heat exchanger operating with the non-uniform inflow of working fluids as a result of using the passive method [16]. It was assumed that an increase in the heat transfer rate could be obtained as a result of forming a stream of gas flowing to the exchanger using metal inserts mounted in the air supply duct. These inserts directed a larger stream of air into the area of a water inlet manifold, where the temperature difference is greatest. The measurements carried out confirmed the validity of this hypothesis.

The present paper deals with the analysis of the impact of the non-uniform but controlled water inflow to a cross-current, tube-and-fin heat exchanger on the effects of its operation. This issue was already preliminarily recognized as part of the doctoral thesis [17], with an indication of potential positive effects, but the author did not indicate how to control the flow of water. The possibility of improving the thermal efficiency of the considered type of heat exchanger by changing the location of the water inlet and outlet nozzles was analyzed in [18].

A separate problem is the mathematical and numerical modeling of heat exchangers which are usually complex devices. Taler and his co-workers have developed finite volume method-based models in simplified [19, 20], and modernized [21] versions. The models enable simulations of different types of heat exchangers (plate, tube-and-fin, including superheaters). A finite volume is however a repetitive segment of a real device. Another option is the use of CFD (computational fluid dynamics) tools, which are also based on the finite volume method. The computing power of modern computers allows to analyze models of whole heat exchangers with a simple structure, but there are still limitations concerning, for example, finned structures. An intensive application of the CFD approach to heat exchangers has been observed since the beginning of the 2000s. The process of designing the heat exchangers is one possible application of this computational approach [22, 23]. Numerical analyses are sometimes limited and they concern one side of a heat exchanger (one heat carrier) [24, 25]. Other investigators utilize the CFD technique for modeling and simulating the media flow maldistribution impact on the heat exchangers performance, as in works [26, 27].

The subject of this study is a cross-flow water-air heat exchanger, the core of which is a bundle of elliptical finned tubes. The work is dedicated

to analyses of the heat exchanger operating under full control distribution of water. The analyses are an extension and supplement to the preliminary tests carried out for the considered heat exchanger, the results of which were presented in the previous work [28]. The results reported there confirmed the possibility of improving the thermal efficiency of a heat exchanger with a non-uniform, but controlled inflow of liquid, postulated by Widziewicz [17]. Earlier studies [17, 28], however, were carried out for only one value of the liquid mass flow rate. The main purpose of the present work is to determine the impact of water distribution between individual tubes for the assumed various mass flow rates of this medium on the thermal efficiency of a heat exchanger. Multivariate analyses were carried out for various temperatures of water and a non-uniform air distribution at the heat exchanger input. The air distribution is a result of measurements. Two variants of water distributions were considered. One distribution was assumed accounting for a non-uniform air inflow, and the second one was calculated for different locations of water inlet and outlet nozzles using the hydraulic model of the heat exchanger. Multivariate simulations were carried out for uniform and non-uniform air inflow using the previously developed numerical model and calculation methodology presented in [29, 30]. The non-uniform air distribution at the heat exchanger inlet is a result of measurements. The obtained results of multivariate simulations were compared with the results of measurement for a case of the non-uniform inflows of heat carriers and with the results of calculations performed under the assumption of uniform flow of both fluids.

2   Measurements

2.1   Characteristics of the analyzed heat exchanger

The heat exchanger, which is the subject of research, is a cross-current, tube-and-fin air-water cooler, the core of which is a bundle of 10 tubes with an elliptical cross-section. The tubes are made of steel and finned with flat, rectangular fins. The view and construction details of this heat exchanger are shown in Fig. 1.

The fins are placed with a pitch of 2.8 mm and there are 175 of them on each tube. The thickness of the tube wall is 1 mm. Hot-dip galvanizing was used to ensure good thermal contact between the fins and the tube surface.
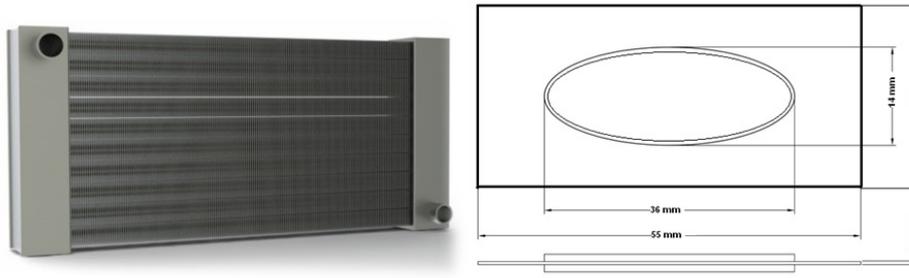
Figure 1: A view and the construction details of the analyzed heat exchanger.

2.2 Test station and method of data analysis

The purpose of the experimental research is, among others, to determine the range and form of a non-uniform airflow through the analyzed heat exchanger, as stated in the introduction. The diagram of the test station is shown in Fig. 2. Its most important element is a computer-controlled hot-wire thermoanemometric sensor, which can be used to measure the distribution of air velocity and temperature at the inlet and outlet of the heat exchanger. Distributions of air velocity and temperature at the inlet to the heat exchanger were input data for numerical analyses in the subsequent stages. For this reason, it is important to ensure the stability of these parameters over time. Velocity and temperature field measurements started after reaching a steady state. Measurements carried out as part of previous studies [10, 11, 17] have proven the repeatability of the results obtained at given air and water flow and temperature settings. In addition, data from
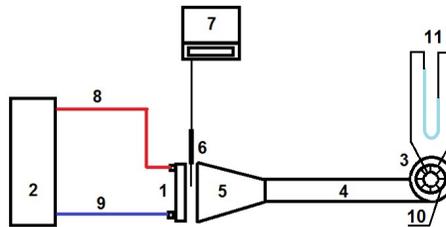

Figure 2: Simplified scheme of the test stand: 1 – heat exchanger, 2 – electric boiler, 3 – fan, 4 – air supply duct, 5 – diffuser, 6 – thermoanemometric sensor, 7 – control computer, 8 – hot water pipeline, 9 – chilled water pipeline, 10 – throttle valve, 11 – manometer.

the hot-wire anemometer sensor are collected at each measurement point in a given time (from a few to several seconds). During this period, sampling takes place with the highest frequency resulting from the data transfer capacity of the measurement card. The results obtained in this way are subjected to statistical processing: outages are eliminated, and the expected value (average) and the standard deviation of this value are determined. The air flow rate is determined by measuring the pressure drop and using the characteristics of the throttle valve, which is used to regulate the gas flow rate.

The station's hot water supply system is equipped with a temperature measurement system (type K thermocouples) and a water flow rate measurement system (a rotameter with a current output). The measurement data allows to calculate the heat flux transferred in the heat exchanger using the equation, expressing a decrease in water enthalpy:

$$\dot{Q}_w = \dot{V}_w \rho_w c_w \left(t_{w,\text{in}} - t_{w,\text{out}} \right), \tag{1}$$

where ρ_w and c_w represent the density and specific heat of water, respectively. The measurements of temperatures of the water at the heat exchanger inlet ($t_{w,\text{in}}$) and outlet ($t_{w,\text{out}}$) and the water volumetric flow rate ($\dot{V}_w$) are considered more accurate than the measurements of air parameters, and thus the thermal performance of heat exchanger was calculated based on presented equation. Uncertainties of the measurements, regarding the heat exchanger output, were calculated using the uncertainty propagation law:

$$u(\dot{Q}_w) = \sqrt{\left(\frac{\partial \dot{Q}_w}{\partial \dot{V}_w} \right)^2 u^2 \left(\dot{V}_w \right) + \left(\frac{\partial \dot{Q}_w}{\partial t_{w,\text{in}}} \right)^2 u^2(t_{w,\text{in}}) + \left(\frac{\partial \dot{Q}_w}{\partial t_{w,\text{out}}} \right)^2 u^2(t_{w,\text{out}})}. \tag{2}$$

The standard uncertainties of measured parameters are based on the resolution of the measuring instruments used. The uncertainties of the water volumetric flow rate measurement and the water temperature measurements (at the inlet and outlet) are 0.6 dm^3/min, 0.06 K, and 0.06 K, respectively.

2.3   Measurement results

As part of this work, nine measurement cases were analyzed. All measurements were carried out at maximum air flow rate, three values of the mass flow rate of the water, and three values of temperature of water behind the boiler 50°C, 70°C, and 90°C. The results of the measurements and

their analysis are presented in Fig. 3 and Table 1 (cases from C1 to C9). The distribution of air velocity at the heat exchanger inlet, presented in Fig. 3, shows very clearly that the inflow of this medium is non-uniform. Differences between maximum and minimum velocity are of up to 400%. As mentioned before, non-uniform air distribution is the data for the calculations.
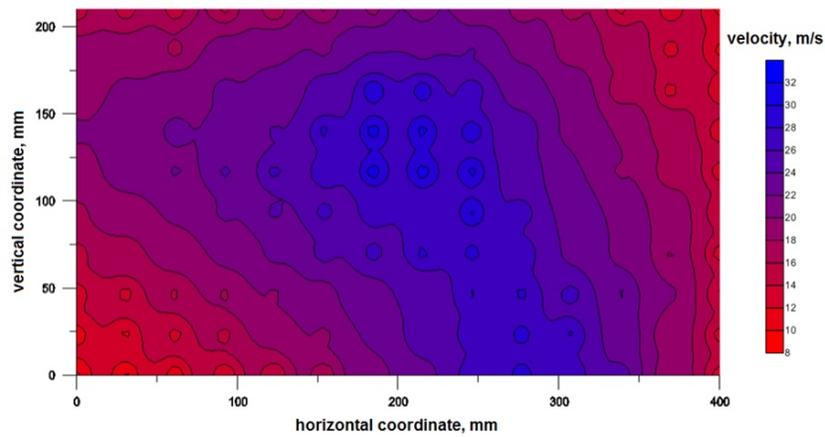

Figure 3: Air inlet velocity distribution measured at the maximum volumetric flow rate (6128 m^3/h).

Table 1: Results of the reference measurements.

Case ID	Air flow rate	Water flow rate	Water temperature		Heat transfer rate	Uncertainty of measurement
			inlet	outlet		
	m^3/h	dm^3/min	$^\circ$C	$^\circ$C	kW	kW
C1	6128	16.7	90.1	75.8	16.66	0.48
C2	6128	16.7	70.2	61.3	10.37	0.29
C3	6128	16.9	49.6	45.3	5.07	0.17
C4	6128	21.8	89.9	78.1	17.95	0.48
C5	6128	21.9	70.5	63.3	11.00	0.34
C6	6128	21.8	49.7	45.9	5.78	0.17
C7	6128	26.5	89.9	79.3	19.60	0.57
C8	6128	26.5	70.1	63.7	11.84	0.33
C9	6128	26.6	50.3	47.1	5.94	0.20

3   Computational models

The computational model of the analyzed heat exchanger was carried out as a part of [17] and was further developed and validated based on the measurement results [16, 29]. The calculation methodology is based on the assumption that the real heat exchanger can be divided into repeatable segments. The heat exchanger operation is simulated by performing sequential calculations for successive repeatable segments.

The division into repeatable segments applied to the analyzed heat exchanger is related to the measurements of the air velocity field at the inlet, during which the air velocity was measured at 140 points. Each exchanger tube was thus divided into 14 sections of equal length, containing 12 fins each. The geometry of the repeatable element is shown in Fig. 4. The element contains the finned tube section, water flowing inside the tube, and a rectangular block of air surrounding the section.
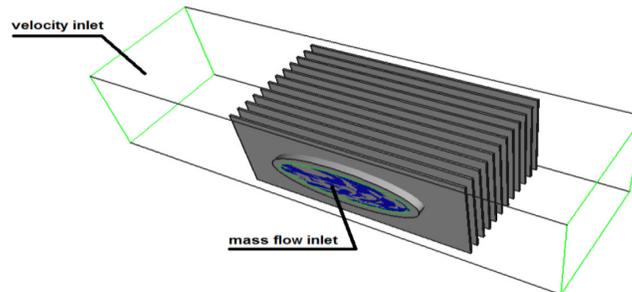

Figure 4: Geometry of the repeatable segment of the analyzed heat exchanger.

The boundary conditions for the first repeatable segment in each tube are the velocity and temperature of air, and mass flow rate and temperature of water at the repeatable element inlet (Fig. 4). It should be stressed here that for the second and remaining repeatable elements of the same tube, the boundary conditions regarding water flow are results of solving the model for previous elements. These conditions are water velocity and temperature fields at the outlet of the element. From the air side, this condition allows taking into account the measurement information about the non-uniform inflow of the fluid, i.e. velocity and temperature of the air inflowing to individual elements. The mass flow rate of water is the result of hydraulic calculations of the heat exchanger in the case of natural, non-uniform water distribution, or it is defined for the case of non-uniform, controlled water

inflow. The remaining planes of the model are considered symmetry planes. The mathematical model consists of a set of equations: flow continuity, energy balance and turbulence model [17, 29, 31]. At the first step, the testing computations aimed at selecting the proper numerical mesh and the turbulence model. The Reynolds stress model of turbulence was selected for basic computations. The developed numerical model was validated based on the measurement results, and the results of this procedure can be found, for example, in works [16, 30].

The geometry of the aforementioned hydraulic model is shown in Fig. 5.
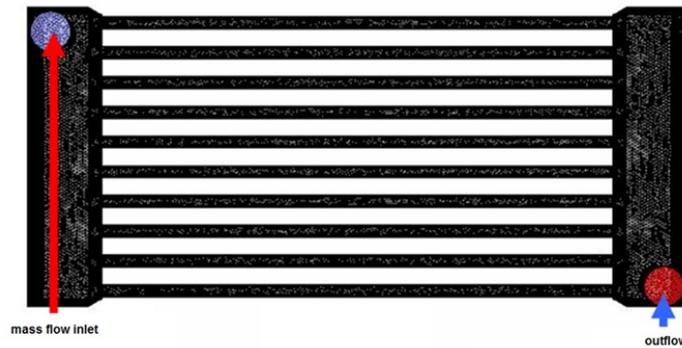

Figure 5: A view of the geometry and boundary conditions of the hydraulic model of the analyzed heat exchanger.

The main boundary conditions are presented in Fig. 5. The mass flow rate of water flowing into the heat exchanger, as well as its temperature, are measured values. The computational domain contains 980 thousand cells which was decided upon the numerical mesh independence test. The test was realized for four discretization schemes: coarse mesh (124 652 cells, mesh number $i = 1$), medium mesh (545 678 cells, $i = 2$), fine mesh (980 128 cells, $i = 3$), and the finest mesh (1 546 258 cells, $i = 4$). The criterion for choice was the relative difference in the water outlet temperature between the subsequent meshes (i and $i + 1$):

$$\Delta = \left| \frac{t_{w,\text{out},i+1} - t_{w,\text{out},i}}{t_{w,\text{out},i}} \right|. \tag{3}$$

The results of the mesh independence test are shown in Fig. 6. It can be seen that making the mesh denser than around 1 million cells does not bring significant changes to the results. Thus, the fine mesh ($i = 3$) was chosen.
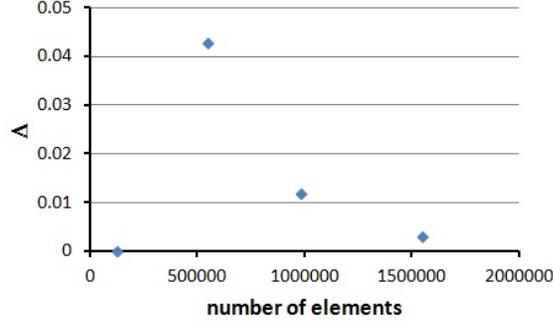
Figure 6: Mesh independence test results.

Testing the turbulence model enabled to choose the k-ε model as giving satisfactory results. First of all, the results given by this model are physically correct. The k-ε model of turbulence is also relatively fast solved. A general form of relationships describing the turbulence kinetic energy k, and its rate of dissipation ε is as follows [31]:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + G_k + G_b$$
$$- \rho\varepsilon - Y_M + S_k, \tag{4}$$

$$\frac{\partial}{\partial t}(\rho\varepsilon) + \frac{\partial}{\partial x_i}(\rho\varepsilon u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial\varepsilon}{\partial x_j}\right] + C_{1\varepsilon}\frac{\varepsilon}{k}(G_k + C_{3\varepsilon}G_b)$$
$$- C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} + S_\varepsilon, \tag{5}$$

where u_i are the components of velocity in x_i-direction, x_i are the Cartesian coordinates ($i = 1, 2, 3$), t is the time, μ_t denotes turbulent viscosity, G_k and G_b represent generation of turbulence kinetic energy due to the mean velocity gradients, and due to buoyancy, respectively. The contribution of the fluctuating dilatation in compressible turbulence to the overall dissipation rate is marked as Y_M. Quantities S_k and S_ε are source terms defined by the user. Turbulent Prandtl numbers for k and ε are denoted by σ_k and σ_ε, respectively. Parameters $C_{1\varepsilon}$, $C_{2\varepsilon}$, and $C_{3\varepsilon}$ are the model constants.

The hydraulic model allowed to determine the distribution of water between the tubes of the analyzed heat exchanger.

4 Results of computations

The numerical model has been solved using the Ansys Fluent software, version 2023R1 [32]. Water has been treated as a Newtonian fluid, with a temperature-dependent density. The solver has been set to a steady time mode. Pressure-velocity coupling of the coupled scheme was applied. Spatial discretization has been set to second order for pressure, second order upwind for momentum, and the first order upwind for turbulent kinetic energy and turbulent dissipation rate. The main results of hydraulic computations of the heat exchanger are the mass flow rates of water flow through particular tubes. As mentioned before, the natural water distribution is an effect of pressure distribution resulting from hydraulic resistances and depending on the shape of the water flow channel. Figure 7 presents the results of computations realized for various values of total mass flow rate at the heat exchanger inlet: 0.4412 kg/s, 0.3622 kg/s, and 0.2785 kg/s, for the location of water inlet and outlet nozzles presented in Fig. 5. This water flow variant was marked as V1.
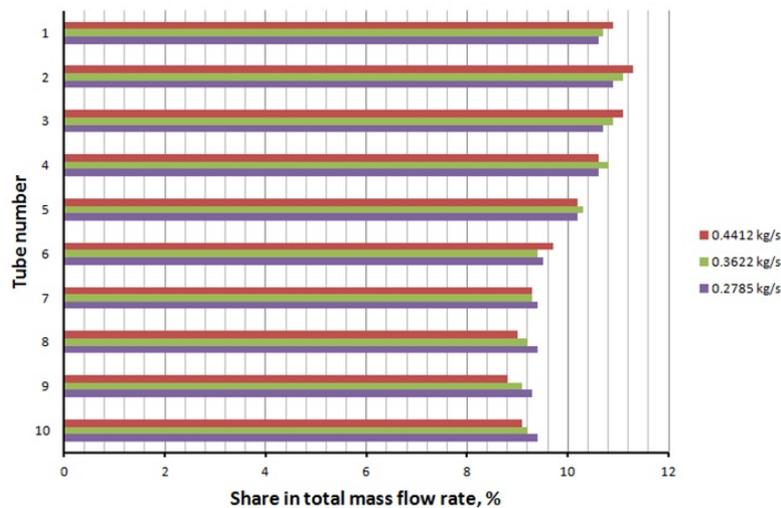

Figure 7: Distribution of water for the non-uniform and uncontrolled flow for three analyzed water flow rates – variant V1.

Assuming the possibility of regulation of water streams flowing through individual heat exchanger tubes, numerical simulations of the heat exchanger operation for two variants were performed. In variant V2, it was assumed that a larger stream of water should flow through the tubes, in the area

in which a larger stream of air also flows. The mass flow of water flowing through a given tube is therefore equal to the proportion of air flowing through the area of the same tube. The input data for this case, presented in Fig. 8, are therefore the result of the assumption made. The distribution of water is independent of the water flow rate as only one air flow rate was considered.
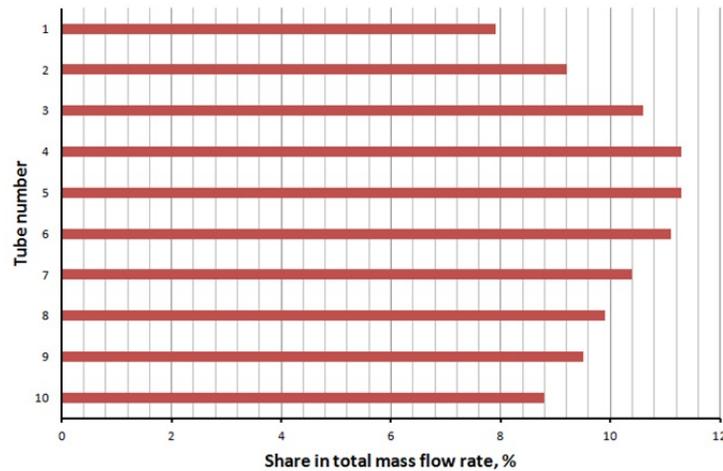

Figure 8: Distribution of water for the non-uniform and controlled flow – variant V2.

The V3 variant assumes that the water inlet and outlet nozzles will be moved towards the center of the height of the water manifolds of the heat exchanger and this would result in symmetrical water distribution relative to the center of the height of the heat exchanger. The location of water nozzles is schematically presented in Fig. 9. This assumption is purely theoretical, however, it refers to the water distribution shown in Fig. 8 to some degree. The values of water mass flow rates for this case are shown in Fig. 10, and they are the result of calculations carried out using the hydraulic model of the heat exchanger. This variant of the water supply cannot be described as controlled, hence it is referred to in the further part of the text by its designation – V3. Some deviation from the symmetric flow of water can be seen there, which may be the result of including gravity in the numerical model.

Analyzing the results of computations done for variants V1 and V3 it may be noted that decreasing the flow rate of water slightly decreases the water distribution non-uniformity. It is more visible for the variant V1.
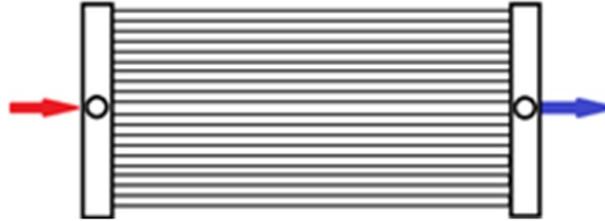
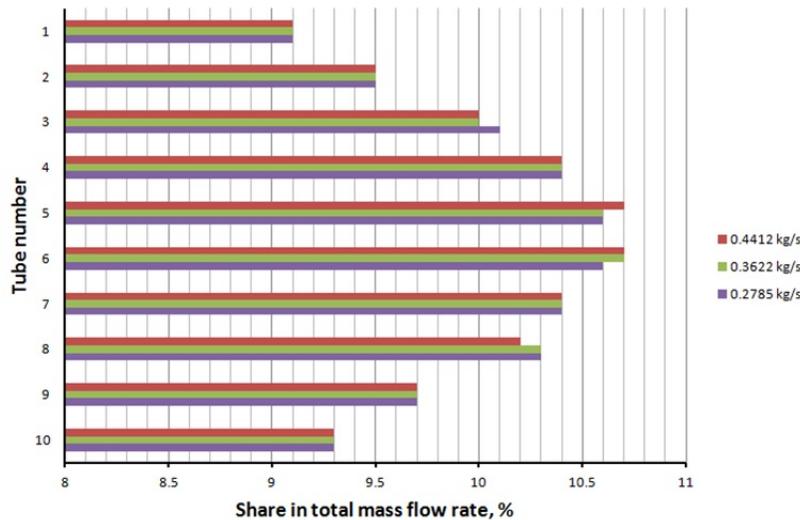Figure 9: Locations of water inlet and outlet nozzles in variant V3.

Figure 10: Distribution of water for three analyzed water flow rates for the variant V3.

Obtaining the uniform flow of the mediums on the test stand for the analyzed heat exchangers is impossible. Therefore, to assess the impact of the non-uniform media flow on the device's performance, measurements were carried out in conditions of the non-uniform inflow of both heat carrier – air and water (variant V1, Tables 2–4) and calculations, using the previously presented model, assuming the uniform inflow of air and water (variant VU, Tables 2–4). The calculations were performed for the aforementioned three values of mass flow rate and temperature of water (Table 1, cases C1–C9). The heat fluxes obtained for the conditions of uniform inflow of the air and the water (variant VU) are 5.56 kW for the set water temperature of 50°C at the minimum water mass flow rate (0.2785 kg/s), and 24.05 kW for the temperature of 90°C at the maximum water mass flow rate (0.4412 kg/s).

This means that the uneven flow of mediums (variant V1) causes a decrease in thermal efficiency of up to 18.5% in the considered cases C1–C9 (Tables 2–4). This decrease in the thermal efficiency of the analyzed heat exchanger is greatest at the highest water temperature at the inlet (cases C1, C4, and C7). It can also be seen by comparing the indicated cases that the increasing water flow rate contributes to a greater decrease in thermal efficiency. The same trend is also seen for the lower inlet water temperature. When evaluating these results, it should be noted that with the increasing flow rate of the hot medium, an increase in the thermal efficiency of the heat exchanger is expected. A direct comparison of thermal efficiency allows this regularity to be noticed. However, the work uses relative values when comparing different variants, and hence the observed effect.

Table 2: Comparison of measurements and computational results – minimum water flow rate

Analyzed case	Flow variant	Heat capacity	Relative heat capacity
		kW	%
C1	Uniform flow of both mediums, VU	19.46	100.0
	Non-uniform uncontrolled flow, V1	16.66	85.6
	Non-uniform flow of air, water, V2	19.64	100.9
	Non-uniform flow of air, water, V3	17.34	89.1
C2	Uniform flow of both mediums, VU	11.54	100.0
	Non-uniform uncontrolled flow, V1	10.37	89.9
	Non-uniform flow of air, water, V2	11.86	102.8
	Non-uniform flow of air, water, V3	10.47	90.7
C3	Uniform flow of both mediums, VU	5.56	100.0
	Non-uniform uncontrolled flow, V1	5.07	91.2
	Non-uniform flow of air, water, V2	5.83	104.8
	Non-uniform flow of air, water, V3	5.26	94.6

Using the previously described numerical model of the heat exchanger and performing calculations for both variants (V2, V3) of water distribution of all analyzed cases, a set of results was obtained, which are presented in Tables 2, 3, and 4. It should be noted that the calculations assumed the non-uniform distribution of air velocity at the heat exchanger inlet (no interference with the natural inflow was assumed), which is shown in Fig. 3.

The results of all analyses, experimental and computational, were compared assuming that for the case of the uniform inflow of both heat carriers, the efficiency of the heat exchanger being the subject of the analysis is 100%.

The summary results presented in Tables 2, 3, and 4 show that the controlled distribution of water (variants V2, V3) in each analyzed case causes an increase in the thermal efficiency of the heat exchanger relative to the non-uniform and uncontrolled inflow of mediums (V1). The results indicate that the inflow of water proportional to the amount of air (V2) allows to achieve even better effect than in the case of the uniform inflow of mediums. This is observed for all considered cases, and the greatest differences from the uniform flow case are observed at the water inlet temperature equal to 50°C (cases C3, C6, and C9). The 4.8% difference was recorded for the C3 case (Table 2). This effect was visible for each water mass flow rate considered, with a slight upward trend with decreasing of water flow.

Table 3: Comparison of measurements and computational results – medium water flow rate.

Analyzed case	Flow variant	Heat capacity	Relative heat capacity
		kW	%
C4	Uniform flow of both mediums, VU	21.52	100.0
	Non-uniform uncontrolled flow, V1	17.95	83.4
	Non-uniform flow of air, water, V2	21.65	100.6
	Non-uniform flow of air, water, V3	18.77	87.2
C5	Uniform flow of both mediums, VU	12.44	100.0
	Non-uniform uncontrolled flow, V1	11.00	88.4
	Non-uniform flow of air, water, V2	12.73	102.3
	Non-uniform flow of air, water, V3	11.02	88.6
C6	Uniform flow of both mediums, VU	6.44	100.0
	Non-uniform uncontrolled flow, V1	5.78	89.8
	Non-uniform flow of air, water, V2	6.70	104.1
	Non-uniform flow of air, water, V3	5.94	92.3

Taking into account the measurement uncertainties, the results may be slightly less optimistic, but they definitely exceed the limits of these uncertainties. The obtained efficiency gains, in the range of 0.2% to 18.6% (calculated relative to the case with the non-uniform and uncontrolled in-

Table 4: Comparison of measurements and computational results – maximum water flow rate.

Analyzed case	Flow variant	Heat capacity	Relative heat capacity
		kW	%
C7	Uniform flow of both mediums, VU	24.05	100.0
	Non-uniform uncontrolled flow, V1	19.60	81.5
	Non-uniform flow of air, water, V2	24.07	100.1
	Non-uniform flow of air, water, V3	20.85	86.7
C8	Uniform flow of both mediums, VU	14.23	100.0
	Non-uniform uncontrolled flow, V1	11.84	83.2
	Non-uniform flow of air, water, V2	14.40	101.2
	Non-uniform flow of air, water, V3	12.58	88.4
C9	Uniform flow of both mediums, VU	6.92	100.0
	Non-uniform uncontrolled flow, V1	5.94	86.3
	Non-uniform flow of air, water, V2	7.08	102.5
	Non-uniform flow of air, water, V3	6.28	90.9

flow V1) are similar to the effects obtained for the case of the controlled air supply, which was tested as part of the work [16]. A comparison of the results obtained for the flow variants V2 and V1 indicates an increase of the thermal output at a level of a dozen percent, and this increase is higher for the highest water inlet temperature. Analysis of results for cases C1, C4, C7, and C2, C5, C8, and then C3, C4, C6 allows to state that better thermal output is observed for the highest water flow rate. The assumption stating that moving the inlet and outlet water nozzles to the half-height of the water manifolds (variant V3) would improve the thermal performance of the heat exchanger under consideration has been also proved. However, in this case, an increase in the thermal output is not as significant as in variant V2, and the results of comparison to variant VU are worse for all the considered cases. The greatest differences are around 5% relative to the V1 variant, and they are observed for cases C7, C8, and C9.

Summing up the performed experimental and numerical analyses, it may be stated that controlling the liquid distribution among the tubes of the heat exchanger seems to be an option for improving the thermal performance of the device.

5 Final remarks and conclusions

A procedure of controlling the flow of water to individual heat exchanger tubes is much more difficult to implement than controlling the air inflow. However, the intention of the authors of the work is not to indicate the technical possibilities, but to analyze the energy effects of controlling the water in cross-flow tube-and-fin heat exchangers operating with non-uniform inflow of working fluids. The results of analyses show that a controlled inflow of water may have a positive impact on the thermal efficiency of the heat exchanger. The rise in efficiency values ranges from 0.2% to 18.6% and from 0.1% to 4.8% in comparison with the natural non-uniform flow of both heat carriers and with the theoretical uniform flow of both mediums, respectively. This rise depends on mass flow rates of water and water temperature at the heat exchanger inlet.

This work is an extension of the earlier authors' research, in which the effects of shaping the inflow (control) of one of the fluids to the considered heat exchanger were checked. The results of the experimental and computational analyses confirmed the earlier observations. The obtained results also confirm the observations of other researchers about the possible positive impact of a non-uniform but properly shaped media inflow on the thermal efficiency of heat exchangers.

The results obtained in this work indicate the possibility of obtaining higher efficiency of heat exchangers in the case of a non-uniform supply of working fluids than in the case of a uniform supply of these fluids. However, finding such possibilities requires, in principle, an individual analysis on a case-by-case basis. For gas-liquid heat exchangers, it seems technically possible to introduce gas supply control. Controlling the flow of liquid on individual heat exchanger tubes is definitely a more difficult task. It is conceivable that such control could be achieved by the installation of a valve system, which would, however, significantly complicate and increase the cost of the overall construction. Another solution could be a specific solution for liquid inlet and outlet manifolds, which, combined with the tube system, would ensure an appropriate distribution of hydraulic resistance. However, the development of such a liquid flow control system requires knowledge of the distribution of the gas stream flowing to the heat exchanger.

Acknowledgements

References

[1] Chiou J.P.: *Thermal performance deterioration in crossflow heat exchanger due to the flow nonuniformity.* J. Heat Transf. **100**(1978), 4, 580–587.

[2] Ranganayakulu Ch., Seetharamu K.N., Sreevatsan K.V.: *The effects of inlet fluid flow nonuniformity on thermal performance and pressure drops in cross-flow plate-fin compact heat exchangers.* Int. J. Heat Mass Trans. **40**(1996), 27–38.

[3] Kumar S., Dwivedi R., Singh P.K.: *Effect of flow maldistribution on thermal performance of water-cooled minichannel heat sink.* In: Advances in Fluid and Thermal Engineering (P. Saha, P.M.V. Subbarao, B.S. Sikarwar, Eds.), 2019, 43–52. doi: 10.1007/978-981-13-6416-7_5

[4] Joseph J.: *Numerical and experimental study of microchannel performance on flow maldistribution.* Micromachines (Basel) **11**(2020), 3, 323, doi: 10.3390/mi11030323

[5] Zhang K., Li M.J., He Y. L.: *A general and rapid method to evaluate the effect of flow maldistribution on the performance of heat exchangers.* Int. J. Therm. Sci. **170**(2021), 107152. doi: 10.1016/j.ijthermalsci.2021.107152

[6] T'joen C., Willockx A., Steeman H.-J., De Paepe M.: *Performance prediction of compact fin-and-tube heat exchangers in maldistributed airflow.* Heat Trans. Eng. **28**(2007), 12, 986–996.

[7] Tereda F.A., Srihari N., Sunden B., Das S.K.: *Experimental investigation on port-to-channel flow maldistribution in plate heat exchangers.* Heat Trans. Eng. **28**(2007), 5, 435–443.

[8] De Schampheleire S., De Kerpel K., Ameel B., Huisseune H., De Paepe, M.: *Thermal analysis of a commercial plate fin heat exchanger with nonuniform inlet flow conditions.* Heat Trans. Eng. **37**(2016), 17, 1466–1477.

[9] Guo J., Huai X., Cheng K., Cui X., Zhang H.: *The effects of nonuniform inlet fluid conditions on crossflow heat exchanger.* Int. J. Heat Mass Trans. **120**(2017), 807–817.

[10] Bury T., Składzień J., Hanuszkiewicz-Drapała M.: *Experimental and numerical analyses of a non-uniform agents flow impact on a finned cross-flow heat exchanger effectiveness.* In: Proc. 22nd Int. Conf. on Efficiency, Cost, Optimization, Simulation and Environmental Impact of Energy Systems – ECOS 2009, Foz do Iguacu, Parana, 31 Aug.–3 Sept. 2009 (on CD).

[11] Bury T., Składzień J., Widziewicz K.: *Experimental and numerical analyses of finned cross flow heat exchangers efficiency under non-uniform gas inlet flow conditions.* Arch. Thermodyn. **31**(2010), 4, 133–144.

[12] Neve R.S., Yan Y.Y.: *Enhancement of heat exchanger performance using combined electrohydrodynamic and passive methods.* Int. J. Heat Fluid Fl. **17**(1996), 4, 403–409.

[13] Coetzee H., Liebenberg L., Meyer J.P.: *Heat transfer and pressure drop characteristics of angled spiraling tape inserts in a heat exchanger annulus.* Heat Trans. Eng. **24**(2003), 6, 29–39.

[14] Bury T., Składzień J.: *Evaluation of selected methods of mitigation of media flow maldistribution impact in finned cross-flow heat exchangers.* Int. J. Thermodyn. **16**(2013), 4, 189-195.

[15] Hajabdollahi Z., Hajabdollahi H., Kim K.C.: *Estimating the non-uniform air velocity distribution for the optimal design of a heat exchanger.* Appl. Therm. Eng. **153**(2019), 704–714.

[16] Bury T., Hanuszkiewicz-Drapała M.: *Numerical and experimental analysis of a tube-and-fin crossflow heat exchanger with controlled non-uniform inflow of media.* E3S Web Conf. **323**(2021), 00005.

[17] Widziewicz K.: *Thermodynamic analysis of a cross-flow heat exchanger considering non-uniformity of working mediums inflow and thermal radiation.* PhD thesis, Institute of Thermal Technology, Silesian University of Technology, Gliwice, 2014 (in Polish).

[18] Bury T., Hanuszkiewicz-Drapała M.: *Analysis of water flow in a cross-current heat exchanger based on a CFD model.* In: Proc. of the 4th Int. Sci. Tech. Conf. on Modern Power Systems and Units, Cracow, 2016, 39–40 (in Polish).

[19] Taler D.: *Mathematical modeling and control of plate fin and tube heat exchangers.* Energ. Conv. Manag. **96**(2015), 452–462.

[20] Węglarz K., Taler D., Taler J.: *New non-iterative method for computation of tubular cross-flow heat exchangers.* Energy **260**(2022), 124955.

[21] Korzeń A., Taler D.: *Numerical modelling of rapid decrease in water flow rate in a plate-and-tube heat exchanger.* Heat Transf. Eng. **44**(2023), 11-12, 1072–1081.

[22] Sunden B.: *Computational fluid dynamic in research and design of heat exchangers.* Heat Trans. Eng. **28**(2007), 11, 898–910.

[23] Bhutta M.M.A., Hayat N., Khan A., Kanwar N.A., Khan S.: *CFD applications in various heat exchangers design: a review.* Appl. Therm. Eng. **32**(2012), 1–12.

[24] Jun S., Puri V.M.: *3D milk-fouling model of plate heat exchangers using computational fluid dynamics.* Int. J. Dairy Tech. **58**(2005), 4, 214–224.

[25] Ozden E., Tari I.: *Shell side CFD analysis of a small shell-and-tube heat exchanger.* Energ. Convers. Manage. **51**(2010), 5, 1004–1014.

[26] Sheik Ismail L., Ranganayakulu C., Shah R.K.: *Numerical study of flow patterns of compact plate-fin heat exchangers and generation of design data for offset and wavy fins.* Int. J. Heat Mass Transf. **52**(2009), 17–18, 3972–3983.

[27] Lee M.S., Li Z., Ling J., Aute V.: *A CFD assisted segmented control volume based heat exchanger model for simulation of air-to-refrigerant heat exchanger with air flow mal-distribution.* Appl. Therm. Eng. **131**(2018), 230–243.

[28] Bury T., Hanuszkiewicz-Drapała M.: *Analysis of a controlled non-uniform liquid inflow impact on cross-flow heat exchanger operation.* In: Advances in the Study of Heat and Mass Transfer (D. Butrymowicz, K. Śmierciew, P. Jakończuk, Eds.), Bialystok Univ. Technol., Białystok 2022, 247–255 (in Polish). doi: 10.24427/978-83-67185-30-1

[29] Hanuszkiewicz-Drapała M., Bury T., Widziewicz K.: *Numerical model of a cross-flow heat exchanger with non-uniform flow of media.* J. Power Technol. **93**(2013), 5, 295–302.

[30] Bury T., Hanuszkiewicz-Drapała M., Widziewicz K.: *The finite difference method and the numerical approach for modeling the maldistribution of the flow of media in tube-and-fin cross-flow heat exchangers.* Heat Trans. Eng. **39**(2018), 13-14, 1179–1191.

[31] ANSYS INC.: *ANSYS Fluent Theory Guide*, Canonsburg, 2021.

[32] ANSYS Fluent Tutorial Guide 2023 R1, http://www.ansys.com (accesed 15 Jan. 2023).