Condensation Of Moist Air In Heat Exchanger Using CFD

Jan Barák, Karel Fraňa, Jörg Stiller

Abstract — This work presents results of moist air condensation in heat exchanger. It describes theoretical knowledge and definition of moist air. Model with geometry of square canal was created for better understanding and postprocessing of condensation phenomena. Different approaches were examined on this model to find suitable software and model. Obtained knowledge was applied to geometry of real heat exchanger and results from experiment were compared with numerical results. One of the goals is to solve this issue without creating any user defined function in the applied code. It also contains summary of knowledge and outlook for future work.

Keywords — condensation, exchanger, experiment, validation.

I. Introduction

SOLVING this problem is part of design optimization of floor convectors used for room cooling. It helps to reduce the energy consumption and improve energy efficiency, which is significant indicator of every energy device. Total annual electricity consumption by air conditioners in Europe is estimated at more than 40 TWh in 2010, two thirds of the total consumption is attributed to the heating function. By 2020 electricity consumption will increase to around 75 TWh annually without any measures (business as usual), mainly due to higher market penetration. The expected increase of 35 TWh annually corresponds to the production of almost five 750 MW-power plants. The planned measures (minimum requirements and energy labels) are expected to lead to savings of around 12 TWh annually by 2020 – only one third of the expected increase in electricity consumption [1].

Condensation of moist air occurs in heated rooms or in floor convectors and it is important for designing of whole exchanger. It affects the construction, because the condensate must be perforce drained away. In moist air presented in the atmosphere water molecules move freely and it sometimes leads to their precipitations. If collision occurs at temperature of dew point or below it, the molecules will not be able to reflex anymore and thus they attach into chains. These molecules are joined into short lines which results into a conversion of water vapour to liquid. If sufficient amount of particles take part in described process then the droplet can be formed [2].

This work is focused to find suitable numerical model to predict the amount of moist air condensate. For better understanding and evaluation of results the model for validation with simple benchmark was developed. The multiphase flow in this problem was simulated by different numerical approaches. Simultaneously, grid and accuracy study was carried out. Obtained knowledge was used for calculation in real heat exchanger. No satisfactory agreement has been found yet, so improved model was created for more suitable results. The aim is to solve this issue using standard implemented models without creating any user defined functions (UDFs).

There are lot of works that applied many customized UDFs to obtain condensation on specified surfaces using authors' own controlled computational program. This makes the problem more difficult, because it is almost impossible to compare used methods and results of their researches. It is not possible to discover the core of their programs, because it is not published in any work. Also, there is less information about condensation than about heating, which was in many works. Sakakura and Yamamoto [3] presented results valid not only for compressible flows but also for flows with low Mach number. This study supposed small amount of moist air in the cooled space. It also presented knowledge about heterogeneous condensation on rudiments. Equation of state is driven by Ishizaka et al. from 1994 and it supposed a mass fraction of created condensate below 0.1. Flow is simplified to two-dimensional case. Computational domain consists of 201x41 elements and has got wall boundary layer. Results were compared with experiment and good agreement was found. The biggest contribution of this work was quantifying the density of particles for creating the most amount of condensate. This density was calculated to be 1E11 particles. Nolte and Mayinger [4] focused on explaining, which parameters affect wall condensation the most. They had confirmed their hypotheses by their own experiments. They found out that heat transfer coefficient is the function of Nusselt's and Sherwood's number. This work does not contain computational fluid dynamics (CFD) calculation. Volchkov, Terekhov V. and Terekhov I. (in 2004) [5] studied general theory about condensation including equations and basic charts. This work contained many results for dependence of heat power on surface area or different criteria numbers on Reynolds number. Whole work is written in theoretic way and does not contain experiments. Saraireh (in 2012) [6] focused on theory of the condensation phenomena. Commercial software FLUENT with many user defined functions (for example on viscosity, thermal conductivity or water density) was used and it led to the excellent agreement with experimental results. Three-dimensional mesh in this case

Jan Barák is with the Power Engineering Department, Technical University in Liberec, Liberec 461 17, Czech Republic, (e-mail: jan.barak@tul.cz).

Karel Fraňa is with the Power Engineering Department, Technical University in Liberec, Liberec 461 17, Czech Republic, (e-mail: karel.frana@tul.cz).

Jörg Stiller is with Institut für Strömungsmechanik, Technische Universität Dresden, Dresden D-01062, Germany, (e-mail: joerg.stiller@tu-dresden.de).

must have over 500 000 elements to be independent on final solution. Usage of polypropylene and aluminium exchanger is being compared. This work contains very detailed mathematic balance description of heat exchanger to capture turbulence effects in the flow. A classical re-normalisation group (RNG) κ-ε turbulence model was used. Very good agreement between numerical and experimental data was found on flows with high Reynolds numbers. Ugurlubilek [7] examined numerical determination of heat transfer coefficient on outer surface of exchanger pipe. Commercial software FLUENT was used and it was supplemented by customized UDFs. Variation of Reynolds number was in range from 28 000 to 230 000. Mesh was tuned by using boundary layer and standard κ-ε turbulence model was used. Good agreement with experiment was observed.

Generally results depend on wide range of parameters, such as: number of nucleation particles inside the gaseous mixture, mass and volume fraction of each phase, temperature difference from dew point, specification of wall's temperature or heat flux, simplified two-dimensional or full three-dimensional computational domain usage, homogeneous or heterogeneous condensation, developed turbulent flow on inlet, wall roughness, quality of environment (cleanliness, regulation and control of relative humidity), radial diffusion of vapour, constructional solution of exchanger etc.

II. TEST CASE

For better understanding of numerical results and effective work progress, was decided to create test case, which will allow examine different program setting. The flow benchmark used only for testing and validation of CFD results was computed with various different numerical models and software, separately. It does not contain part of removing the condensate out of computational domain.

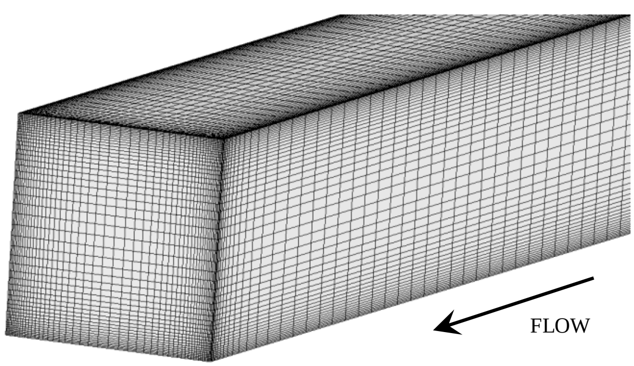


Fig. 1: Computational mesh

Fig. 1 shows a cut of computational domain. It represents flow in square channel with dimensions of 50 mm x 50 mm x 1000 mm and bias factor on edges. Only quadrilateral elements were used with edge sizing parameter. The flow is assumed to steam axially with the constant speed on inlet surface of 0.1 $m \cdot s^{-1}$ and inlet temperature of 295.15 K in atmospheric pressure of 101 325 Pa. Constant temperature of 274 K was considered for longitudinal walls. This temperature was determined in order to be under the dew point, which is 284.7 K. Simultaneously it is required to avoid temperature below the freezing point. Velocity was specified as flow parameter on inlet area. Inlet relative humidity is considered to be 51.6%, because it corresponds with the relative humidity while making the experiment. Inlet Reynolds number is calculated using standard equation:

$$Re = \frac{v_{in} \cdot L}{\nu} \tag{1}$$

where v_{in} is inlet velocity [$m \cdot s^{-1}$], L characteristic dimension [m] and υ kinematic viscosity [$m^2 \cdot s^{-1}$]. Inlet Reynolds number is 330, calculated using (1). Mass fraction of water vapour is calculated using equation:

$$\omega_{wv} = 1 - \frac{1}{1+SH} \tag{2}$$

where ω_{wv} is mass fraction of water vapour in mixture [$kg_{vapour} \cdot kg_{mixture}^{-1}$] and SH is specific humidity [$kg_{vapour} \cdot kg_{dry\_air}^{-1}$]. This leads to mass fraction of water vapour 0.00842, calculated using (2). Volume fraction of water vapour is calculated using equation:

$$\Phi_v = 1 - \frac{m_{da}}{\rho_{da}} \cdot \frac{\rho_{ma}}{m_{ma}} \tag{3}$$

where Φ_v is volume fraction of water vapour in mixture [$m_{vapour}^3 \cdot m_{mixture}^{-3}$], m_{da} mass flow of dry air [$kg_{dry\_air} \cdot s^{-1}$], ρ_{ma} density of moist air [$kg_{moist\_air} \cdot m^{-3}$], ρ_{da} density of dry air [$kg_{dry\_air} \cdot m^{-3}$], and m_{ma} mass flow of moist air [$kg_{moist\_air} \cdot s^{-1}$]. Volume fraction of water vapor is 0.00229, calculated using (3). These values are necessary for correct input into calculation software, where CFX computes with mass fraction of each phase and others with volume fraction of each phase.

Laminar and turbulent boundary layers are treated differently in terms of the condensation mass flux at the surface. For laminar flow, the condensable mass flux is [8]:

$$\dot{m}_{c,lam} = \frac{W_B}{W_M} \frac{D_{BM} \cdot \rho_m}{\delta} ln\left(\frac{1-x_B(\delta)}{1-x_B(0)}\right) \tag{4}$$

where $m_{c,lam}$ is condensation mass flux [$kg \cdot m^{-2} \cdot s^{-1}$], W_B and W_M is the molecular weight of the condensable component and of the mixture component (of condensable and non-condensable) respectively [$g \cdot mol^{-1}$], D_{BM} is the binary diffusion coefficient [-], ρ_m density of mixture [$kg \cdot m^{-3}$], δ is the height of the boundary layer [m] and x is molar fraction[-]. For a turbulent boundary layer, the condensable mass flux is calculated as shown in (5) [8]:

$$\dot{m}_{c,turb} = -T_M \frac{Y_{BP} - Y_{Bw}}{1 - Y_{Bw}} \tag{5}$$

where $m_{c,turb}$ is condensable mass flux [$kg \cdot m^{-2} \cdot s^{-1}$], T_M is wall multiplier decided by the form of the turbulent wall function, Y_{BP} is the mass fraction of the condensable component at a grid point near the wall [-] and Y_{Bw} denotes the mass fraction of the condensable component at wall [-].

One of tested computational models was Volume of Fluid. It is a surface-tracking technique applied to fixed Eulerian mesh. This model was proposed by Hirt and Nichols in 1981. It was designed for two or more immiscible fluids, where the position of the interface between the fluids is of interest. The fluids share a single set of momentum equations, and the volume fraction of each of the fluids in each computational cell is tracked throughout the domain [9]. Volume of Fluid preserved constant volume fraction of air, but did not compute any condensate. Another tested model was Mixture. It differs from the Volume of fluid model in three respects: the Mixture model allows the phases to be interpenetrating and allows to move at different velocities using the concept of slip velocities and there is interaction of the inter-phase mass, momentum and energy transfer in the mixture model [9]. Mixture did not preserve constant volume fraction of dry air in the computational volume and did not produce any condensate. That is why it is not suitable for this work. Improved Mixture wall film is also not suitable for the same reasons. So called Eulerian model is the most complex model of all tested ones. Its great advantage is in computing enthalpy for each of the phases in the mixture, so is suitable for phase change models. It was not able to start a calculation so results cannot be presented. Wet Steam model (density-based solver) is specially developed for vapour flows. It also contains equations for calculating with nucleation. Although this model seems to be the most suitable for this case, it was not able to start calculation, because this model computes with one phase vapour only. In this case it is necessarily to compute with mixture of water vapour and dry air. CFX in version 14.5 contains module of wall condensation. Software preserved constant mass fraction of air and also was able to calculate condensed water vapour. This model was found to be the most suitable for calculations like this. Two tests of the multiphase flow with condensation process were considered. First test presented case, where there was zero heat transfer through walls, so they had exactly the same temperature as surrounding. It is assumed that mass balance must be conserved. Convergence was more rapid, mass fractions of phases stood still and no condensate was created. Second test run with dry air only, it means zero mass fraction of water vapour inside. Temperature of walls was the same as default (274 K). In this case no condensation occurred in computational domain. These two tests were successful, because models proofed all theory hypotheses. These two tests showed pertinence of this approach, so CFX will be used for solving this application. The grid study independence was also done and results for created condensate were without changes only when mesh had more than 200 000 elements. It was decided that final mesh will have around 250 000 elements.

A. Analytical Determination of Condensate

Amount of condensate calculated analytically was determined by using Moliere chart assuming thermodynamic equilibrium at outlet area. For precision calculations of equations for moist air properties the Microsoft Excel program was created. The difference of absolute humidity on inlet and outlet areas was multiplied with mass flow of dry air, so maximal amount of condensate could be specified. The mass flow was 0.00029829 kilograms of dry air per second. It can be assumed, that moist air on outlet has 100% relative humidity, because CFX has calculated average temperature on outlet area of 275.2 K. This almost corresponds with boundary condition for longitudinal wall (274 K), so equilibrium could be assumed. Convective and diffusive effects in boundary layer are not considered. Multiphase flow with condensation was simulated as three-dimensional steady state flow.

B. CFD Results

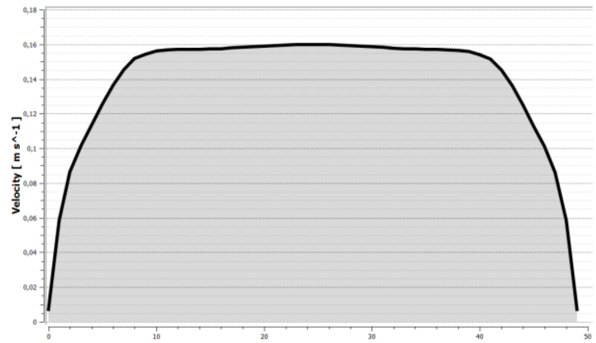


Fig. 2: The velocity profile at outlet

Velocity profile of flow at outlet is on fig. 2. Profile is not yet parabolic, because the flow is not yet fully developed.

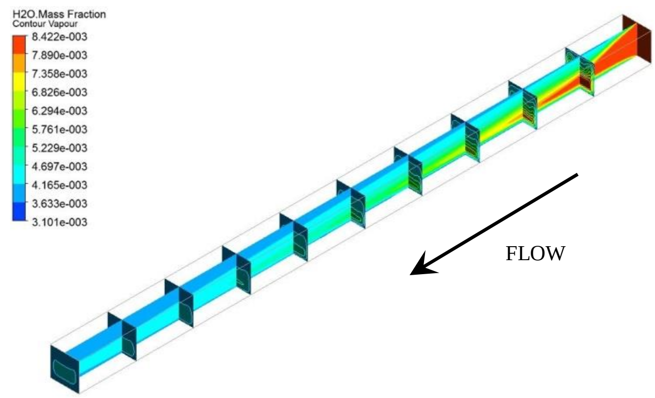


Fig. 3: Contours of water vapour mass fraction

Fig. 3 depicts the change of water vapour mass fraction inside computational domain. It corresponds with theory and shows, that on inlet is the highest mass fraction of water vapour and at outlet area is the lowest. Differences represent creation of condensate. The most of condensate is generated in the second half part of the channel, because flow is there cooled under dew point.

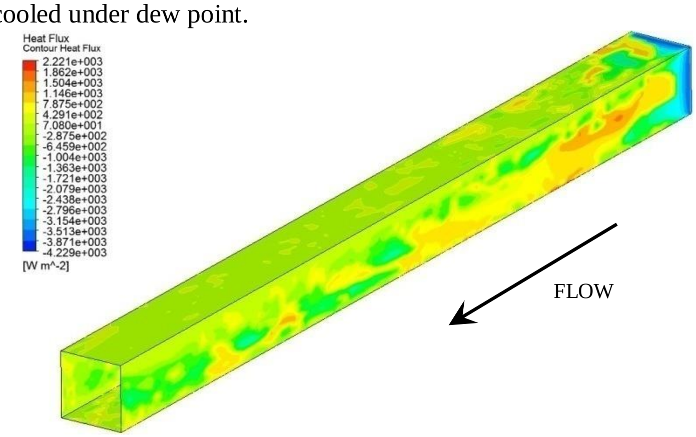


Fig. 4: Heat flux at longitudinal walls

Fig. 4 depicts heat flux on longitudinal walls calculated by CFX. Area average value is -98.68 $W \cdot m^{-2}$. It is difficult to determine heat transfer coefficient, because during dropwise condensation the heat transfer coefficient can be more than 10 times larger than during film condensation [10].

Generally, results are symmetrical, because of the absence of gravity in the numerical model. It should be mentioned that CFX analysis does not contain implemented nucleation model, therefore it is necessary to create customized UDFs for more accurate calculation results and for control the growth of droplets, or if higher accuracy is required.

C. Comparison of Results

TABLE I
COMPARISON OF RESULTS

Title	Value [$g_{vapour} \cdot hour^{-1}$]
Calculated by CFX	4.536
Calculated analytically	4.3856

The difference between amount of condensate determined by software and analytically is 3.43 %. This is good agreement; however is the simple flow benchmark. Nevertheless, the successful numerical test could confirm that used multiphase models, with others required settings, was found to be appropriate for such numerical simulation problems.

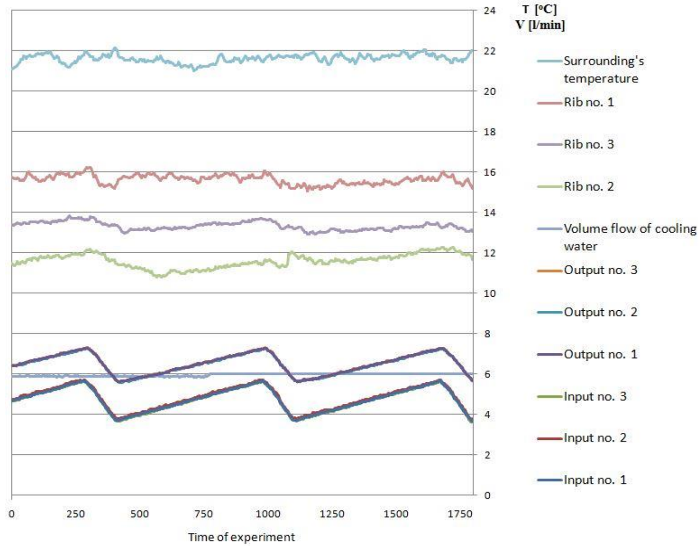
III. EXPERIMENT

The selected model was applied to the real exchanger. This exchanger is tested and measured in laboratory in cooperation with company Licon Heat Ltd. The aim of this experiment is to measure amount of condensate under controlled conditions. Experimental results were compared with numerical results to obtain rigorous model of exchanger for easy prediction of changes for exchangers designed and created in future.

A. Experiment Description

Fig. 5: Exchanger in laboratory

Fig. 5 shows assembly of exchanger, fan and humidity sensor. On left side there are connected pipes from and to chiller, which generates cold supply water. Before starting the experiment, whole assembly was running for more than 80 minutes to reach steady state. After that time the experiment continued without any changes, but with new receptacle for collecting the condensate. This receptacle is not visible, because it is situated under the exchanger. Experiment run for exactly 3600 seconds and after it was done, it was measured the difference of mass between full and empty receptacle.



Ch. 1: Conditions during experiment (cut)

Chart no.1 captures a record of conditions during the experiment. It shows temperature on ribs measured using thermocouples, surroundings temperature measured by thermometer and volume flow of cooling water measured by inductive flow sensor.

TABLE II
DIFFERENCE OF MASS OF RECEPTACLE

Title	Value [grams]
Receptacle with condensate	563.675
Empty receptacle	299.512

Table II shows growth of mass (which is the amount of created condensate) during the experiment. It can be calculated that 264.163 grams of condensate per hour was created. This value is necessary for comparing it with numerical results obtained further.

B. Numerical Results

Aim of this part is to create real exchanger model and to compare its results with experimental ones. Only the space between two ribs was considered and translation boundary conditions were used on corresponding areas. The real exchanger contains exactly 234 ribs, so obtained amount of condensate must be multiplied with this number. Computational mesh used for calculations is tree-dimensional. Special attention was focused on making the boundary layer in area near tubes [11]. It contains of 180 959 elements. Results are valid for RNG κ-ε turbulent model with standard 5% intensity of turbulence and turned on the gravity force. The RNG approach was used. It is a mathematical technique that can be used to derive a turbulence model similar to the κ-ε, results in a modified form of the ε equation, which attempts to account for the different scales of motion through changes to the production term [12]. Inlet Reynolds number is 15 560, calculated using (1).

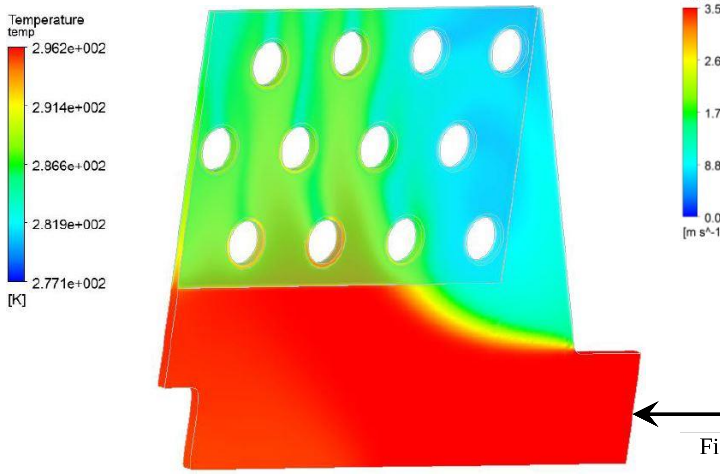


Fig. 6: Temperature field

Fig. 6 depicts temperature field in computational domain. It corresponds to the space, where moist air occurs in heat exchanger between two ribs. Figure demonstrates influence of cooling, where red colour stands for high temperature (especially at inlet) and light blue colour represents cooled moist air and area where the most of condensate is created. For closer idea, what happens inside the computational domain, it would be helpful to show results in the plane, which is situated precisely in the middle of computational domain.

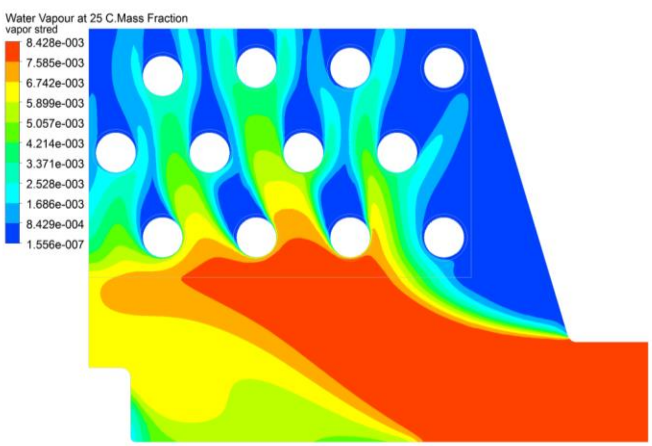


Fig. 7: Water vapour contours in middle plane

Fig. 7 depicts computed water vapour field in space between two ribs in ideal exchanger. Real exchanger differs in the shape of ribs - they may be for example warped. The amount of water vapour fraction is decreasing because of condensation, where blue colour indicates area with less water vapour in flow. The most intensive condensation appears in the space where blue colour is indicated. This is region with lower velocity, so that there is enough time for condensation process.

Fig. 8 describes velocity field and explains turbulence differences inside heat exchanger. It may be helpful to consider the change of location of each pipe in case of designing new or updated heat exchanger.

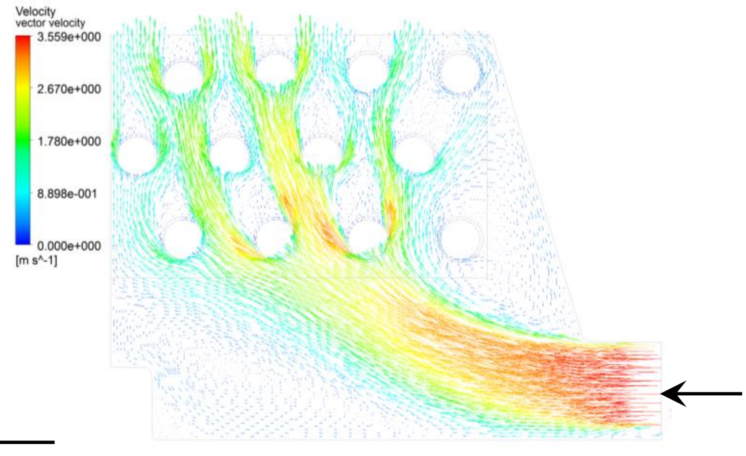


Fig. 8: Velocity field in middle plane

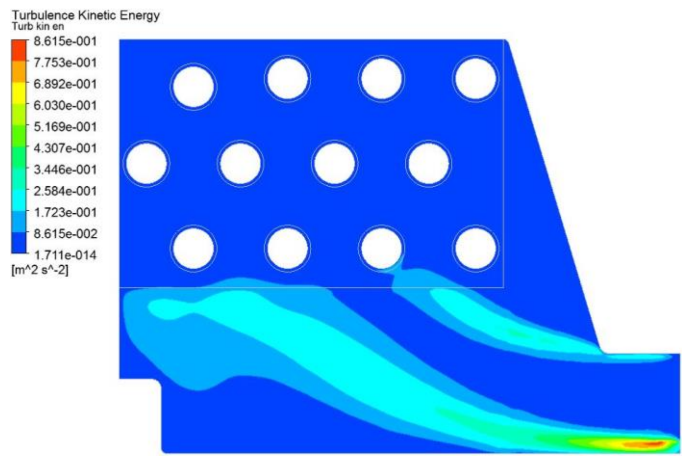


Fig. 9: Modelled turbulence kinetic energy

Fig. 9 describes the modelled turbulence kinetic energy, where RNG κ-ε turbulent model was used. It is the most significant on periphery of the highest speed.

TABLE III
COMPARISON OF RESULTS

Source	Value	Percentage comparison
Experiment	264.163	100%
Numerical model	766.836	290%

Table III contains comparison of experimental and numerical results of created condensate. It is obvious, that numerical model calculated almost three times more condensate. This over-prediction can be explained with many factors, such as: model is designed with straight ribs and not with another shape, which influences parameter called efficiency of rib and it has big effect on heat transfer coefficient; model has assigned constant temperature on ribs, but in reality there is complicated temperature field; model has constant velocity on inlet area from fan and it does not respect outlet velocity profile from fan; in model is no further definition of solid-fluid interface on rib. This knowledge leads to the statement, that improved model of exchanger must be developed for higher accuracy of prediction, including more accurate specification of parameters mentioned in this paper.

IV. CONCLUSION

This work is focused on theoretical description of condensation and especially on its computational modelling for prediction of created condensate. To find functional model, another model with simple geometry of square canal was created. Elementary boundary conditions were used to obtain easily-reproducible results. This model was tested with different computational codes and computational models. Good agreement was found by using CFX software only. Two tests were run to proof the stability and correct calculation. First case considered the same temperature of walls with surrounding and second one flow with dry air only. Model passed these tests and proofed suitability of using CFX software in cases like this. This knowledge was used on real exchanger model, which is tested and measured in laboratory under controlled conditions. Experiment was run and this paper contains description and results of it. RNG κ-ε turbulent model with 5% intensity turbulence on inlet area was used. Comparison of numerical and experimental results showed difficulties using simple boundary conditions in computational model. It was found out that computational model over-predicted amount of condensate almost three times. This work contains the list of parameters, which affect condensation more or less and how to deal with them. Improving the accuracy could be the aim of next work on this project. For more accurate control of amount of condensate, it was decided to create another model with user defined functions for number of condensate particles inside the calculation domain with respect to cleanliness of environment. It should be mentioned, that temperature field on rib with specified shape has great influence on efficiency of cooling. A big advantage of this work is, that functional model for condensation was found. It also presents summary of knowledge about condensation phenomena modelling.

ACKNOWLEDGMENT

This work was made possible by the grant TA01020231 “An applied research focused on the increase of the thermal efficiency of the heat exchanger and confirmation of the service conditions in relation to the renewable energy sources” provided by the Czech Technological Agency. This work was also supported by European Project no CZ.1.07/2.3.00/20.0139 “Building of an excellent scientific team necessary for experimental and numerical modeling of fluid mechanics and thermodynamics”.

REFERENCES

[1] A. Michel, E. Bush, J. Nipkow, C. Brunner, H. Bo, “Room air conditioners: Recommendation for policy design”, Paris, 2012, available online: http://www.topten.eu/uploads/File/Room%20air%20conditioners%20Recommendations_May%202012.pdf.

[2] Condensation, available on Merriam Webster at: http://www.merriam-webster.com/dictionary/condensation.

[3] K. Sakakura, S. Yamamoto, “Numerical and experimental predictions of heterogeneous condensate flow of moist air in cooled pie” in International Journal of Heat and Fluid Flow, 2006, vol. 27, issue 2, pp. 220-228. DOI: 10.1016/j.ijheatfluidflow.2005.08.006.

[4] G. Nolte, F. Mayinger, “Condensation from steam-air mixtures in a horizontal annular flow channel”. Experimental Thermal and Fluid Science, 1988, pp. 373-384.

[5] E.P. Volchkov, V.V. Terekhov, V.I. Terekhov, “A numerical study of boundary-layer heat and mass transfer in a forced flow of humid air with surface steam condensation” in International Journal of Heat and Mass Transfer, 2004, vol. 47, 6-7, pp. 1473-1481. DOI: 10.1016/j.ijheatmasstransfer.2003.09.018.

[6] M. Saraireh, “Heat transfer and condensation of water vapour from humid air in compact heat exchangers”, Victoria University, 2012, Dissertation Thesis, School of Engineering and Science.

[7] N. Ugurlubilek, “Numerical estimation of the condensate flow rate on the condenser pipe”, Journal of Engineering and Architecture, Faculty of Eskisehir Osmangazi University, 2011, XXIV, no.2.

[8] M. Lejon, “Wall condensation modeling in convective flow”, KTH Industrial Engineering and Management, Master of Science Thesis, 2013

[9] A. Eghbalzadeh, M. Javan, “Comparison of mixture and VOF models for numerical simulation of air-entrainment in skimming flow over stepped spillways”, 2012 International Conference on Modern Hydraulic Engineering, doi:10.1016/j.proeng.2012.01.786

[10] P. Talukdar, "Condensation", study literature for students, Department of Mechanical Enginnering, IIT Delhi

[11] T. B. Gatski, M.Y. Hussaini, J.L. Lumley, “Simulation and modeling of turbulent flows”, 1996, New York, Oxford University Press

[12] M. J. Izadi, J. Ammouie, “Numerical analysis effect of sweep angle and taper ratio on lift and drag coefficients of a cessna wing”, Ninth International Congress of Fluid Dynamics and Porpulsion, 18-21th December 2008, Alexandria, Egypt