19th International Conference on Modelling and Applied Simulation
17th International Multidisciplinary Modeling & Simulation Multiconference

ISSN 2724-0037 ISBN 978-88-85741-48-5 © 2020 The Authors.
DOI: 10.46354/i3m.2020.mas.007

Humid Air Condensation in Heat Exchanger

Jan Barak[1,*], Karel Frana[1]

[1]Technical University of Liberec, Faculty of Mechanical Engineering, Department of Power Engineering Equipment, Studentská 2, Liberec, 461 17, Czech Republic

[*]Corresponding author. Email address: jan.barak@tul.cz

Abstract

Paper is devoted to humid air condensation in water-air heat exchangers. It presents thermal comfort parameters including dependency on dew point of humid air inside room. Main target is to depict numerical model of humid air condensation with its initial and boundary conditions which was used to determine amount of condensate under constant operating conditions. Paper also mentions experiment designed by authors with same geometry and operating conditions like numerical model. Comparison of results gained by those two independent methods are shown and the most critical points of interest for phase change models are highlighted.

Keywords: thermal comfort, humidity, dew, condensation, cfx

1. Introduction

Studies show that people spend more than 90 % of their time in interiors. Ability to monitor, manage and improve human environment is crucial because it directly influences health, well-being and work efficiency of occupants in interiors. Quality of air is complex multidisciplinary field. Many input parameters must be taken into consideration for design of healthy and comfortable surroundings. One of significant characteristic is distribution of humidity and temperature in rooms which can be jointly adapted using for example air conditions or floor heat exchangers.

1.1. Composition of humid air

Humid air in interiors may contain many various pollutants depending on purpose of room, used building materials, geological position, exterior pollution from an industry or transportation etc. They negatively influence perceived well-being and associated comfort. These pollutants may be of chemical origin (radon, carbon dioxide, ozone, tobacco smoke, volatile organic compounds, odor etc.), biological origin (bacteria, mold, legionella etc.) or solid non-biological particles like dust, asbestos, lead, garbage or moisture. Hazard of some substances is based on their invisibility and difficult identification. Negative effects are mostly detected after long-term exposition which hampers to fast elimination of risks.

Solution for improvement of humid air is in detecting source of hazardous compound leak, its elimination and mainly in well managed heat, ventilation and air conditioning systems during long-term maintenance. High quality of breathing air without pollutants is basic human right for everyone.

Interconnection of many parameters impact how do occupants feel inside room. Commonly used term "comfort and feeling" according to literature contains for example temperature, temperature difference between head and feet, air circulation, humidity distribution, light dissolution, material of things, color scheme, noise level, present of decoration and plants. There is direct link between temperature of dew point and comfort perceived by occupants, which is not so well-known fact. For calculations of dew point is necessary to know exact temperature and humidity.

1.2. Goals and methods for this work

Above mentioned facts defined main target of this paper which is condensation of humid air, especially quantify amount of condensate. This topic has been studied on water-air heat exchanger which is being installed under floor of rooms as alternative to air conditioning. In studied case exchanger is accompanied with fan and protective case in box. Typical place of installation is in front of a window. Fan sucks air from room and blows it between ribs of exchanger. Cooling water circulates in tubes of exchanger and when it is cold enough (it cools ribs of exchanger under dew point of humid air), condensation occurs on ribs which leads to cooling and drying of humid air. This paper describes numerical model which was used to quantify amount of condensate and allowed to study phase change flow in narrow slip represented by ribs of exchanger. Results of numerical solution are being compared with self-designed experiment with identical initial and boundary conditions. Experiment is not described in this work in detail. Numerical model which would predict well flow variables and heat transfer calculations would be very useful in wide range of industry fields because it may offer digital prototyping with shorter time for new product development.

2. Literature review

This chapter describes mainly theoretical knowledge published by other scientists for heat exchangers, humid air and results of their work. Generally, less work is presented for cooling processes than for heating processes.

2.1. Heat exchanger design

When heat exchanger is used as heating device there are no significant difficulties during its operation. Problems occur when heat exchanger is intended to be used as cooling device. Increased technical requirements lead to slightly different heat exchanger design. It is necessary to ensure suitable joining of corrosion-free materials at their interface and also necessary spacing between ribs since condensate droplets reduce flow cross section and create heat transfer barrier. Studied exchanger has two copper tubes and aluminum ribs which are connected by cold forming. Those materials are typical representatives for such exchangers because they offer good match between heat transfer rates, economic and production site. Case of heat exchanger assembly must also contain part for drain of condensate.

2.2. Fluid flow – Humid air

One and only flow component in studied case is humid air which is ideally mixture of dry air and superheated steam only. In reality different pollutants act like condensation nuclei and create instabilities inside flow. Pollutants will not be considered in this work because they may disproportionately complicate studied case.

It is necessary to fully understand thermophysical properties of humid air and analyze their mutual connections. No license-free software was found to study humid air properties on academical level so authors of this paper created sophisticated function in MATLAB software for precise calculations. Input parameters are: temperature, humidity and pressure. Function precisely calculates 38 variables. Source code can be found in Barak (2019) and is free of charge.

2.3. Publication analysis

Literature survey for condensation of humid air indicate that not many authors are focused on condensation in comparison to heating processes. Many theoretical works were done long time ago where main focus was to describe condensation phenomena for film condensation on vertical wall by governing equations analytically. Recent works include numerical part by computational fluid dynamics (CFD) but they did not show satisfying results so far mainly because of convergence difficulties. Most of the work was done using user-defined functions as supplement for CFD code to reach good agreement with corresponding experiment or theory. This situation complicates comparison of results to find suitable solution which is good for wide industry applications. Ideal solution would be to find suitable software with implemented phase change models which are well described in associated documentation.

Generally, there are two main groups where condensation could occur. First is on solid walls where film or droplets can be formed depending on surface properties, heat rate and mainly time because droplet condensation always becomes thin film on wall after certain amount time of exposition. Second group contains so called spontaneous condensation where droplets are created in volume of air on condensation nuclei which could be for example any pollutant or foreign object. Only droplet condensation on solid walls is considered for this work because it offers the highest rate of heat transfer. Regime of condensation is influenced by many parameters, for example material, wettability of its surface, heat transfer rate, wall orientations etc.

Ugurlubilek (2011) numerically studied heat transfer coefficient on outer surface of aluminum tube which was exposed to ambient environment. Difficulties for convergency were observed. Commercial software with user-defined functions was used which led to great agreement with selected references. Work determines dependency of heat transfer coefficient on exchange surface area, temperature difference and latent heat.

Karkoszka (2007) described in detail physical and mathematical models with related equations for film condensation on walls, different boundary conditions for natural and forced convection.

Saraireh (2012) described experiment of condensation in polypropylene exchanger with corresponding numerical model where many user-defined functions were used. One part of work was validated with experiment and showed excellent agreement but another part used results of other researchers to validate results and no satisfactory agreement was observed in this case. Studied regime was film condensation. Simplified model was observed to be unsatisfactory and author recommends to use three-dimensional model of computational mesh. Polypropylene exchanger seems to be equivalent with those from non-ferrous materials.

Sakakura (2006) numerically investigated condensation of humid air in horizontal tube where second half of length was cooled with various heat exchange value. Inlet conditions were constant and two-dimensional mesh of ca 8 000 nodes was used. Numerical model used many user-defined functions and results were compared with experiment designed by author. The most satisfying results were observed when humid air contained 10^{11} vapor particles in cubic meter of humid air.

Deponti (2011) described humid air condensation in automotive front lights. In this case flow is always laminar, surrounding faces have high temperature with intensive heat exchange including radiation absorption, emission and reflexing effects. Test chamber allows to control temperature and humidity of surrounding of light and thus can simulate different conditions on road. Commercial software ANSYS CFX was used with 1 750 000 elements of three-dimensional computational mesh. High sensitivity for boundary and initial condition was observed. Most of condensate was created of places with small magnitude of velocity and temperature. Numerical model can predict amount of condensate which is not the case during experiment.

Zschaeck (2014) with colleagues studied the condensation of steam on the vertical wall of a nuclear reactor as one of the naturally passive mechanisms for cooling the reactor in the event of a refrigerant leak. Their work focuses on the mathematical description of condensation in turbulent flow in CFD and validates it with two laboratory tasks. Steam was used as the flowing medium as a condensing element and air as a non-condensing gas. The work describes the relationship between wall functions, mass flow of substances and molar fractions of humid air components. The wall temperature in calculations is isothermal and the model contains the interface of the solid with the flowing medium. The three-dimensional problem was simplified to a two-dimensional computing mesh that contained up to 784 000 nodes. It was found that the computational mesh with 220 000 nodes is sufficiently fine and independent on numerical solution. The calculations were defined as stationary and in all of them the turbulent model k-ω SST by Menter was used with automatic calculations of flow at the walls. For simplicity, the model included a plane of symmetry. The gas was defined as ideal with constant values of density and molecular diffusivity. The turbulence intensity was 5 % and the ambient pressure was atmospheric. An important finding is that the model overestimated the amount of condensate in all cases, so it was necessary to use the condensation coefficient into the model, which was 0,3. It was sufficient for the calculation that the residues were $1 \cdot 10^{-4}$ so that the error was at most 0,06%. Although the development trend corresponded to the measured data, it was found that the computer software underestimated the formation of condensate and the heat transfer coefficient by about 20%. When applying this assignment to one of the validation models, the condensation intensity factor had to be reduced to 0,1. It has been found that increasing the turbulence intensity at the inlet into the computational region and increasing the ratio between turbulent viscosity and molecular dynamic viscosity will increase condensate production and heat flux during inter-substance transfer.

Volchkov (2004) presented summary of theoretical knowledge of heat and mass transfer inside boundary layer during condensation phenomenon. Variability of Prandtl, Schmidt and Lewis numbers as functions of Reynold's number are discussed because of uneven distribution of temperature and concentration components. His theoretical work is compared with experimental results of other scientists.

Grooten (2011) in detail describes experiment where growth of droplets was examined. Heat exchanger was made of polyvinylidene fluoride. Formation of droplets was captured by high-speed infrared camera. Temperature, humidity, flow properties and mass increment of condensate were very frequently logged and after certain amount of time droplets were torn down by silicone string. First important output specifies that growth of droplets follows power law where base is time of exposition and exponent is mass fraction of water vapor contained in humid air. Second output describes that the overall heat transfer resistance decreases as the droplet removal rate increases.

Parihar (2014) focused on mathematical description of condensation. Heat transfer coefficients are set for film and dropwise condensation. Material and surface roughness are highlighted as important parameters affecting mechanism of creation condensate from nuclei. It was observed that heat transfer coefficient decreases with increasing absolute surface roughness and with increasing height of condensate. Authors emphasize necessity to take into account nuclei distribution.

2.4. Summary

Second chapter focuses on humid air definition and presents main outputs from previous work of other scientists in history. It can be declared that theoretical knowledge about humid air condensation is comprehensive including governing equations. These equations must count with phase change, latent heat with re-evaporation and regime of condensation. Numerical modeling of condensation meets unsatisfying results so far because of many tailor-made solutions designed for specific cases which leads to incomparability of results.

3. Numerical model of humid air condensation

Elementary motivation for creating numerical model of condensation is to decrease time of new exchanger development and increase associated cost savings. When model will be enough accurate and precise in terms of air flow and heat transfer it would be possible to study internal flow with heat exchange in detail which may be sometimes impossible during experiment. However, as literature review shows, no universal setting for all exchangers has been found yet because of high complexity of heat transfer including phase change models. This chapter depicts numerical model developed by authors to study humid air condensation in floor heat exchanger.

3.1. Geometry

Because heat exchanger is geometrically simple device with many repeating sections it seems to be beneficial to use implemented software functions and model only one section with corresponding periodic boundary conditions. Figure 1 depicts geometry of numerical model which corresponds to heat exchanger assembly in reality. Dark grey color represents aluminum ribs and shiny color represents copper tubes. Their connection is strong and tight due to cold forming production process. Translation boundary conditions are defined in middle of gap between two adjacent ribs. Because of asymmetrical design of heat exchanger, it was decided to study effect of rotation of heat exchanger with respect to direction of flow. Only functional part with ribs was considered for numerical model thus no endings are modelled. Section of heat exchanger is inserted into rectangular channel to get defined volume where humid air flows, from right to left. Directly under heat exchanger, there is a hole where condensate drops into a collecting vessel for its weight measurement.

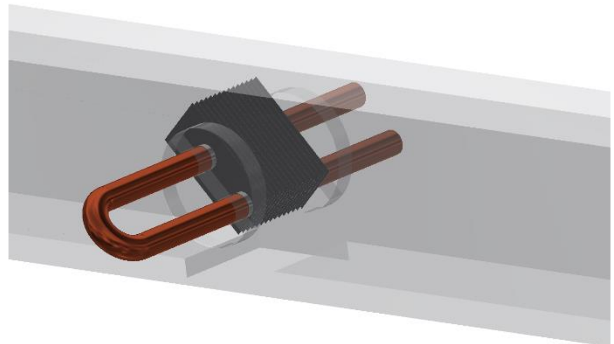

Figure 1. Geometry of exchanger assembly

3.2. Computational Mesh

Generally computational mesh must correspond with flow regime to obtain high quality elements especially near wall. In studied case turbulence model was k-ω SST which requires y+ values around 1 or below to be able to predict heat transfer without wall treatment. This requirement significantly shapes size and style of mesh. Refined mesh for studied case is depicted on Figure 2. For total volume of $22 \cdot 10^{-6}$ m^3 was discovered that mesh needs to have minimally 29,7 million elements to be independent on final solution. Only such small elements have chance to predict rise of condensation nuclei. Therefore, high performance computing was necessary to calculate case. Green color on left side of Figure 2 represents copper tube and grey color with yellow part represent aluminum ribs. Biggest volume between aluminum ribs is volume where humid air flows. It is necessary to model boundary layer at connection between solid materials and fluid domain with conformal mesh to avoid numerical approximations. Mesh is stationary, average quality of elements is 0,82.

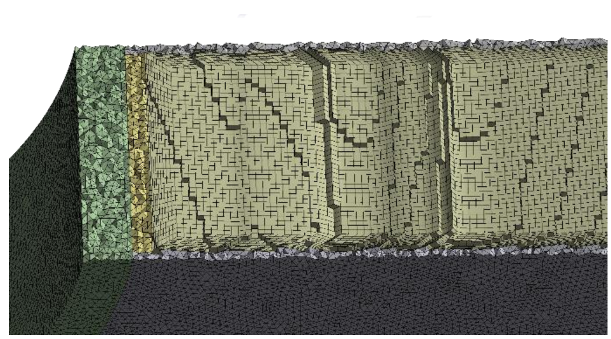

Figure 2. Computational mesh

3.3. Preprocessing

This section describes setting used in software ANSYS CFX 18 to achieve humid air condensation. It may be used as tutorial because no work like this was found.

At first it is necessary to define materials. Ideally humid air consists of ideal dry air and superheated water steam only. In reality some impurities may occur, see section 1.1, but they will not be taken into account in this work. First user-defined material must be of so called "Variable Composition Mixture" from "Air Ideal Gas" and "Water Vapor at 25°C" which results into conserving mixture of those components with variable mass fractions of components. Second user-defined material represents creation of condensate from water steam and is defined as "Homogeneous Binary Mixture" where first material is "Water Vapor at 25°C" and second material "Water" as condensate. In this material it is very important to carefully specify its saturation properties because it significantly affects amount of created condensate in controlled volume.

Next step is defining domains in controlled volume. In presented case there are solid materials represented by copper tubes and aluminum ribs, both are solid with constant density and enabled heat transfer. Humid air flow volume is modelled using fluid domain with humid air inside (Air Ideal Gas as Constraint and Water Vapor at 25 degrees Celsius with specified kinematic diffusivity and condensation model enabled on interface between fluid and solid domains), specified ambient pressure, buoyancy model, turbulence model (in this case Shear Stress Transport k-ω) with heat transfer on.

Selected setting results into creation of two domain interfaces, one for solid-solid connections and second for solid-fluid connections. Both of them must have conservative interface flux turned on and General Grid Interface (GGI) mesh connection method.

Whole computational domain with symbols of boundary conditions is shown on Figure 3 where application of periodical boundary condition is visible on first sight. In studied case simple exchanger with only two tubes was used to reach geometrical simplicity. One of important parameters for managing efficiency is angle between direction of flow and center line of exchanger, which is 90 degrees on Figure 3. Boundary conditions on inlet represents definition of velocity profile with temperature field, mass fraction, turbulence kinetic energy and eddy frequency specification. Mass fraction of water vapor in humid air is very small, i.e. $5 \cdot 10^{-3}$ and its precise calculation may be found in Barak (2019). Pressure outlet or Opening were specified on outlet area even if this combination (mainly with velocity inlet) generally does not lead to satisfactory fast convergence. Inner cylindrical surface of tubes has defined constant temperature given from experiments and it must necessarily be under dew point of humid air and at the same time must allow flow of cooling medium, in this case water.

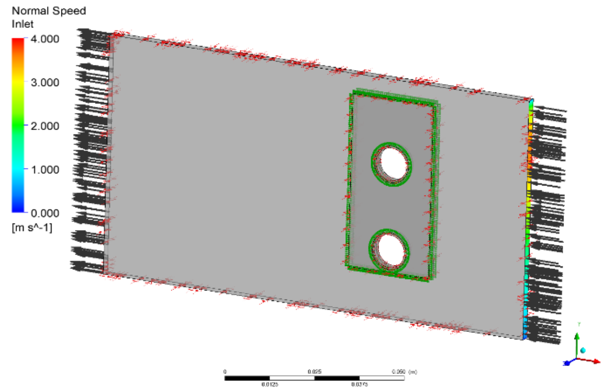

Figure 3. Boundary conditions

Calculations were run using High Resolution Advection Scheme with RMS Residual Convergence Criteria set for $1 \cdot 10^{-4}$ which demonstrates problems to reach convergence in general for this type of calculations.

3.4. Processing

Described case was calculated using workstation with 12 core processor with 96 GB RAM and it took around 1 400 core hours to reach one task for one angle of attack of exchanger.

3.5. Postprocessing

One of many advantages of CFD is ability to study flow in places where conventional experimental methods do not allow to insert their sensor. As example could be used narrow chink between two aluminum ribs which has distance only 2,7 millimeters. Central plane in middle of described chink was chosen to be representative plane on which different variables will be shown. Velocity field in middle plane is shown on Figure 4.

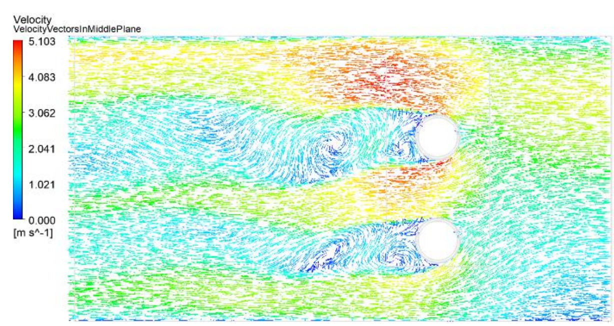

Figure 4. Velocity field in middle plane of exchanger

On right side is defined velocity profile from boundary conditions, after flow is being affected by heat exchanger geometry where the biggest influence is generated by two tubes of exchanger. In heat exchanger there is slight aerodynamic clogging because of smaller area of cross section. Eddies generated by tubes are visible behind cylinders and they influence temperature field as shown on Figure 5.

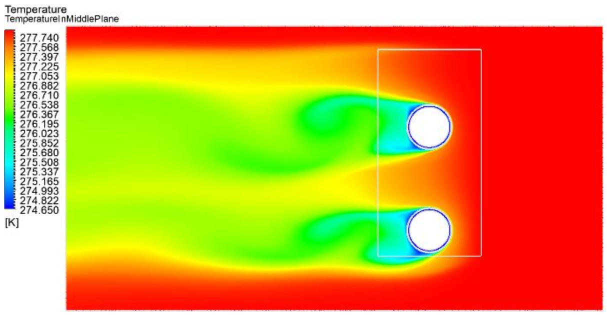

Figure 5. Temperature field in middle plane of exchanger

Figure 6 shows volume where ANSYS CFX predicted creation of condensate. As expected, condensate will be mainly created on aluminum ribs (interface between solid and fluid domain with lowest local temperature). One can observe that part of ribs which is far from tube (on top on Figure 6) is being heated by flow and not enough cooled by negative heat transfer from cooling water inside copper tubes. Effect of flow direction can be observed as well.

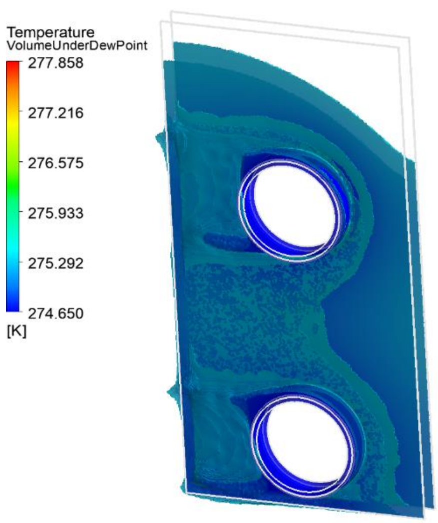

Figure 6. Predicted volume of condensate

Figure 7 shows calculation of Y+ value which represents local quality of computational mesh and must correspond with used turbulence model and heat transfer model. In this case the most sensitive area is on external surface of tubes where the value around 1 or below must be present to get correct results.

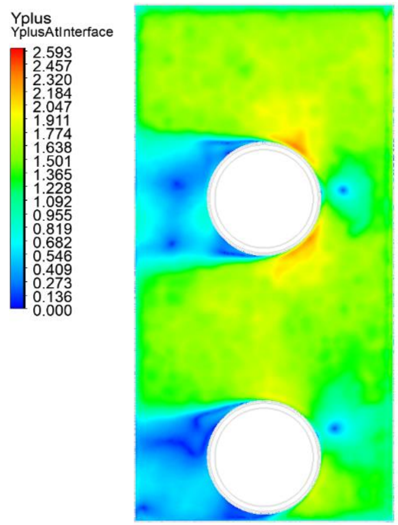

Figure 7. Y+ value in critical area

4. Results

Results of calculations are shown on Figure 8 with orange color. Horizontal axis represents angle of attack (rotation of exchanger from flow direction) and vertical axis represents amount of condensate created in time. Blue line represents results from experiment which was run with same boundary and initial conditions as numerical calculation. It is visible on first sight that numerical model overpredicted amount of condensate always minimally twice but trend was same. This overprediction may indicate systematic mistake in numerical model so it is important to find root cause of such an overprediction.

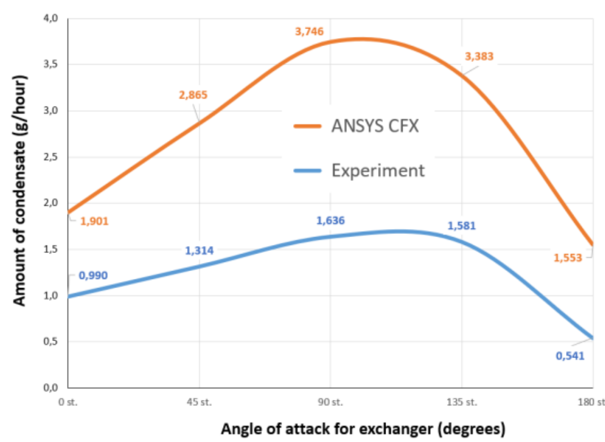

Figure 8. Comparison of results

Sensitivity study was done to find which parameters influence amount of created condensate by numerical model the most. Following list names main areas where improvement can be found and also corresponding results:

- Quality of computational mesh – test for mesh independence on numerical solution was done so final fine mesh is precise enough including limits for aspect ratio, skewness, and orthogonal quality;

- Boundary conditions – would influence general task and would not be aligned with experiment so no changes are possible;
- Initial conditions – Must correspond with experiment as well as boundary conditions;
- Solid domains definition – Are set by physical properties of materials, no change is necessary;
- Fluid domain definition – Definition of ideal humid air was presented in this paper and impurities are not considered. It was found out that significant changes can be observed on material with "Homogeneous Binary Mixture" setting which represents phase change from water vapor into water. There are many options in ANSYS CFX how to define saturation properties for phase change material like general (specification of pressure and temperature), by table, as real gas using different Kwong (or Peng Robinson model) or by IAPWS (International Association for the Properties of Water and Steam) Library. It was observed that value of saturation pressure has biggest effect on amount of created condensate. It corresponds with theory for local decrease of pressure in place where droplet is being created;
- Interface between domains – Indicate correct setting using conformal mesh;
- Check of implemented condensation model in software documentation – ANSYS implemented only film condensation into its software and not droplet or spontaneous condensation (only for wet steam) which may explain difference in results as well.

Unsatisfying difference of results between CFX model and experiment may be also explained by mistake during measurement of condensate in experiment. It was observed that droplets on aluminum ribs were still present during measurement which indicates that steady state conditions of condensation were not reached yet. Different amount and size of droplets on wall is only initial phase to steady state status, which is thin film of condensate on wall. It means that new experiment run is necessary with longer time of run (measurement was done after 3 hours and it was not enough).

Overprediction of numerical model may be also found in Schramm (2012), Zanzi (2018) and Barak (2013).

4.1. Summary of knowledge

Third chapter of this work presented created numerical model for humid air condensation in detail. It may be useful for those who need to correctly specify phase change model in commercial ANSYS CFX software because no such tutorial has been found by Barak (2013) in academic literature. Concept of chapter is sequential as model is being created. Critical and important points of work flow are highlighted and commented. Main output is that standard implementation of CFX calculates with film condensation and thus overpredicts amount of condensate for droplet regime of condensation.

5. Conclusion

This paper is devoted to humid air condensation in heat exchanger. It describes necessity to solve this problem on academic and industry level because humidity and associated effects significantly affect quality of air in all buildings all around the world. Paper presents humid air in general with link to precisely calculate its thermophysical properties, heat exchanger design, outputs from literature survey and previous results reached by other scientists. The most important part of work is detailed description of numerical model of heat exchanger which was developed and its results were compared with experiment sharing same boundary and initial conditions. Results are presented, differences are discussed and explained. Numerical model overpredicted amount of condensate minimally twice in all cases comparing to experiment. This is not yet acceptable to be applied on industry level so further study is necessary and improvements are outlined. New ideas about measurement of condensate are presented as well so new run of experiment is necessary to get more accurate amount of condensate.

Acknowledgments

This work was supported by the Ministry of Education, Youth and Sports of the Czech Republic and the European Union - European Structural and Investment Funds in the frames of Operational Programme Research Development and Education - project Hybrid Materials for Hierarchical Structures (HyHi, Reg. No. CZ.02.1.01/0.0/16-019/0000843).

References

Barak J., 2019. Studie vlivu vnějších a vnitřních podmínek na kvalitu prostředí v místnostech a budovách. Dissertation thesis. Technical University of Liberec.

Barak J., 2013. Kondenzace atmosférické vlhkosti ve výměníku tepla pomocí CFD. Diploma thesis. Technical University of Liberec.

Deponti A., Damiani F., Brugali L., Bucchieri L., Zattoni S., Alaimo J., 2011. Modelling of condensate formation and disposal inside an automotive headlamp.

Grooten M.H.M., 2011. Dropwise condensation from flowing air–steam mixtures: Diffusion resistance assessed by controlled drainage. International Journal of Heat and Mass Transfer 2011 54: 4507-4517.

Karkoszka K., 2007. Mechanistic modeling of water vapour condensation in presence of noncondensable gases. Doctoral thesis. KTH Engineering Sciences.

Parihar S.S., 2014. Mathematical Modeling on Dropwise Condensation. International Journal of Current Engineering and Scientific Research 2014: 30-35.

Sakakura K., Yamamoto S., 2006. Numerical and experimental predictions of heterogeneous condensate flow of moist air in cooled pipe. International Journal of Heat and Fluid Flow 27:220-228.

Saraireh M., 2012. Heat Transfer and Condensation of Water Vapour From Humid Air in Compact Heat Exchangers. Dissertation thesis. Victoria University Melbourne.

Schramm B., Stewering J., Sonnenkalb M., 2012. Validation of a simple condensation model for simulation of gas distributions in containments with CFX. Computational Fluid Dynamics (CFD) for Nuclear Reactor Safety Applications - Workshop Proceedings. Experimental Validation and Application of CFD and CMFD Codes to Nuclear Reactor Safety Issues.

Ugurlubilek N., 2011. Numerical Estimation of the Condensate Flow Rate on the Condenser Pipe. Journal of Engineering and Architecture Faculty of Eskisehir Osmangazi University 24:41-49.

Volchkov E.P., 2004. A numerical study of boundary-layer heat and mass transfer in a forced flow of humid air with surface steam condensation. International Journal of Heat and Mass Transfer 2004, 47: 1473-1481.

Zanzi S., 2018. Numerical Simulation of Condensing CO2 Supersonic Flows at High Pressure. Diploma thesis. Politecnico di Milano.

Zschaeck G., 2014. CFD modelling and validation of wall condensation in the presence of non-condensable gases. Nuclear Engineering and Design 279: 137-146.