

# Mathematical model of compact type evaporator

Martin Borovička<sup>1,\*</sup>, Tomáš Hyhlík<sup>1</sup>

<sup>1</sup>CTU in Prague, Department of Fluid Mechanics and Thermodynamics, Czech Republic

**Abstract.** In this paper, development of the mathematical model for evaporator used in heat pump circuits is covered, with focus on air dehumidification application. Main target of this ad-hoc numerical model is to simulate heat and mass transfer in evaporator for prescribed inlet conditions and different geometrical parameters. Simplified 2D mathematical model is developed in MATLAB SW. Solvers for multiple heat and mass transfer problems - plate surface temperature, condensate film temperature, local heat and mass transfer coefficients, refrigerant temperature distribution, humid air enthalpy change are included as subprocedures of this model. An automatic procedure of data transfer is developed in order to use results of MATLAB model in more complex simulation within commercial CFD code. In the end, Proper Orthogonal Decomposition (POD) method is introduced and implemented into MATLAB model.

## 1 Introduction

An increasing effort to decrease the cost of any device requires more advanced methods for precise simulation and modeling of physical phenomena. Heat and mass transfer in heat exchangers/evaporator with complex geometry is still a complicated problem, because an analytical solution is not often available, so numerical simulation along with measurement is the only option that can be used for heat duty estimation and geometry optimization. In this paper, focus is on modeling heat and mass transfer inside evaporators for air dehumidification. Selected evaporator for this application is classified as a cross flow compact type heat exchanger (or finned tube heat exchanger) - tube bundle (single tube with multiple passes) with common fins (every rectangular fin is in contact with all passes of the tube), using R134a as a refrigerant. An evaporator (“Inside Heat exchanger” in Fig. 1.) is placed in heat pump circuit that can be often found in many air dehumidification devices. Other parts of heat pump circuits are: condenser, compressor and capillary tube/reduction valve.

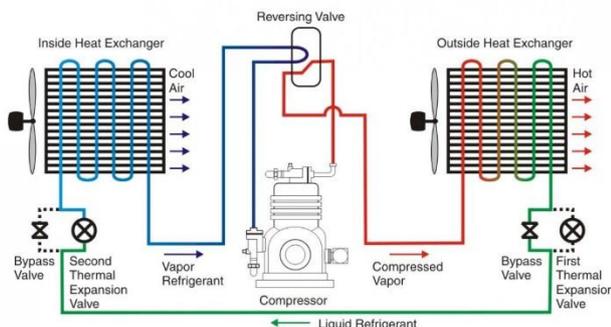


Fig. 1. Heat pump circuit [1]

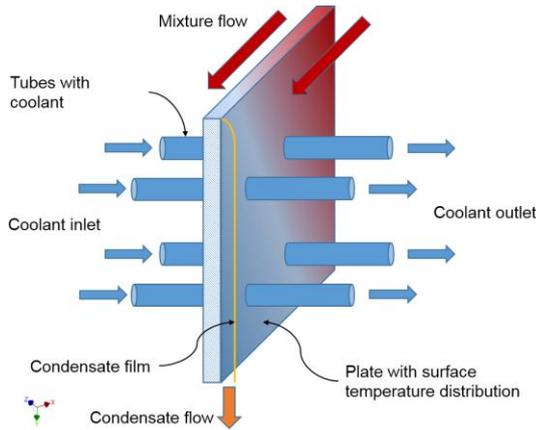
## 2 Mathematical model

The mathematical model is developed in MATLAB SW, because of user-friendly environment, simple implementation of matrix operations and ability to use advanced optimization tools for model validation and development. The model contains simple menu interface for user convenience and tool for data transfer into commercial CFD code. Optimization of code has been performed so model can be used for optimization of heat exchanger geometry.

### 2.1 Model properties

The mathematical model simulates heat transfer between humid air and coolant/refrigerant flowing through tubes. Next, simulates mass transfer on outer side - water vapor condensation from air into water film on fins and tubes, and refrigerant evaporation on internal (tube) side. The model predicts locations, where condensation occurs, based on local flow parameters and local fin/tube temperature. Model is parametric in inlet conditions (air/refrigerant temperature and mass flow, humidity, pressure) and also in heat exchanger geometrical parameters. A semi-automatic procedure, that will manage data transfer of results from the MATLAB model into the simulation in commercial CFD package, using scripts and macros is also developed and implemented. A very important attribute of the model is its dimension – in how many axis parameter/property distribution is solved. Choice of model dimension and its complexity affects precision, computation time and computational resources. To obtain a balance between model precision, computation time and computation requirements, 2D model is used. 2 dimensions (coordinate  $x$  and  $y$ ) where flow parameters are solved correspond to 2 edges of a fin as shown in Fig. 2.

\* Corresponding author: [Martin.Borovicka@fs.cvut.cz](mailto:Martin.Borovicka@fs.cvut.cz)

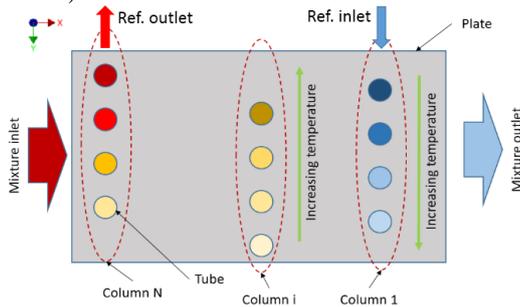


**Fig. 2.** 2D model schema

From point of view of data transfer into commercial CFD, 2D model is not the best, because in remaining, not simulated coordinate, all parameters are constant. This fact might cause imprecise results if big non-uniformities in this coordinate are present – for example non-uniform velocity in inlet caused by inlet duct.

## 2.2 Coordinate system and flow orientation

Heat exchanger with arbitrary dimensions and its coordinate system is in **Fig. 3**. ( $x$  – length,  $y$  – height,  $z$  – width)



**Fig. 3.** Coordinate system orientation

Air flow is always orientated in  $x$  direction (from left to right), and refrigerant flow is oriented in  $z$  direction – “inlet tube section” is placed at upper right corner of fin and “outlet tube section” at upper left corner (only even number of tube columns with staggered pattern is supported in this version).

## 2.3 Simplifying assumptions

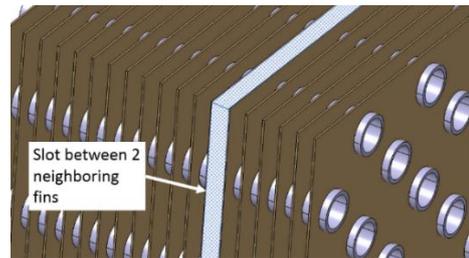
Modeling heat and mass transfer is complicated geometry is a complex problem, so couple simplifications are assumed: [2]

- 1) The fin is flat. (corrugated fins are usually present, the effect of corrugation is included in  $AreaFx$ ,  $AreaFy$  parameters – area increase factors)
- 2) For the first guess, refrigerant temperature is constant (equal to boiling temperature) through whole evaporator. During simulation, refrigerant temperature distribution is developing.
- 3) Condensate from tubes is dropping down, not flowing down the fin.
- 4) Temperature of fin is constant through its thickness.

- 5) No heat flow from fin edges.
- 6) All fields of flow parameters (temperature, velocity, humidity ...) are constant in  $z$  direction in every slit.
- 7) Steady-state flow.
- 8) Perfect contact between outer tube surface and fin.

## 2.4 Governing equations

Differential equations describing heat and mass transfer problems are listed below. Equations (1) and (2) describe the general balance of mass for air and film flow. Ad hoc equation (3), (4) and (5) describe conservation of enthalpy in 1 slot (space between 2 neighboring fins). Discretization of the equation (6) is used for a surface temperature analysis.



**Fig. 4.** Flow volume region

- Continuity equation for air can be written as

$$\frac{\partial \rho_a}{\partial t} + \frac{\partial (\rho_a u_{a,i})}{\partial x_i} = 0. \quad (1)$$

- Continuity equation for water film is

$$\frac{\partial \rho_f}{\partial t} + \frac{\partial (\rho_f u_{f,i})}{\partial y} = 0. \quad (2)$$

- Enthalpy conservation equation for humid air is

$$dQ_{a \rightarrow f} = 2q_{a \rightarrow f} dx dy = dH_a - V_a dp_a. \quad (3)$$

- Enthalpy conservation equation for water film is

$$-dQ_{f \rightarrow w} + dQ_{a \rightarrow f} = (-q_{f \rightarrow w} + q_{a \rightarrow f}) dx dy = dH_f. \quad (4)$$

- Enthalpy conservation equation for refrigerant/coolant can be written as

$$dQ_{w \rightarrow c} = dH_c - V_c dp_c. \quad (5)$$

- Fourier’s law for heat convection is

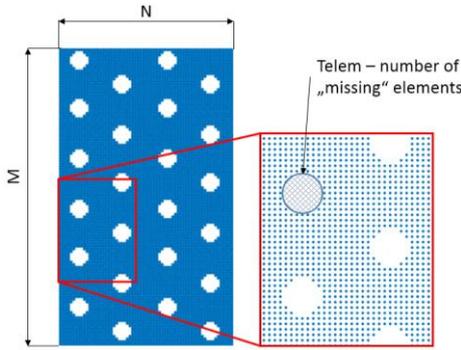
$$\vec{q} = -\lambda \cdot \nabla T. \quad (6)$$

- Equation of state for ideal gas is

$$p = \rho RT \quad (7)$$

## 2.5 Meshing

Automatic mesh generation (and tube bundle generation) on fin is included in the model. (only parameter of the mesh is  $Telem$  - #of elements per tube cross-section). Number of elements in  $x$  direction -  $N$  and  $y$  direction -  $M$  is solved automatically for given  $Telem$  parameter by sub-optimization (minimization) of tube cross section area error -  $ErrSurf$ .



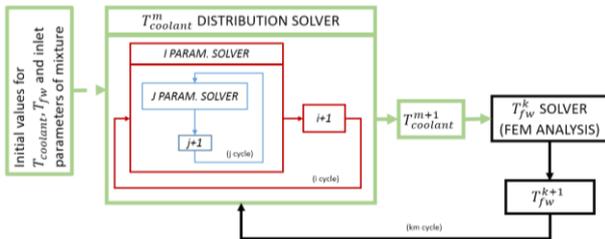
**Fig. 5.** Volume mesh

For numerical simulation, it is necessary to divide tube into elements/segments in which temperature/enthalpy increments are evaluated. In this model, tube element is defined by tube segment between 2 neighboring fins.

## 2.6 Solver sub-procedures

MATLAB model itself is composed of multiple sub-procedures, which solve various heat and mass transfer problems. [2]

- Surface temperature solver
- Refrigerant temperature solver
- “i + j” cycles to solve air and film local parameters (Air/film enthalpy solver)
- Film thickness solver
- Film temperature solver



**Fig. 6.** Solver schema

The iterative solution procedure is obvious from the diagram above. First, the initial distribution of air temperature and condensate flow field is solved from guessed refrigerant temperature distribution. Based on new air temperature, a new refrigerant temperature distribution is solved, from which new fin surface temperature is evaluated. New refrigerant and surface temperature are then used to solve for new air temperature and condensate flow and iteration loop is completed. A number of iterations required to obtain stable solution depends on inlet conditions, but varies from 50 to 150.

### 2.6.1 Film thickness solver

Because the focus is on evaporators for air dehumidification, low condensate flow is expected, thus, small thickness of a film is assumed. Velocity gradient for a simple 1D case of fluid film on vertical wall can be then derived as [3] [4]

$$\frac{du_f}{dz} = \frac{\rho_f - \rho_g}{\mu_l} g(\delta - z) + \frac{\tau_g}{\mu_l}. \quad (8)$$

Ordinary differential equation above is integrated and boundary conditions applied, velocity profile is obtained.

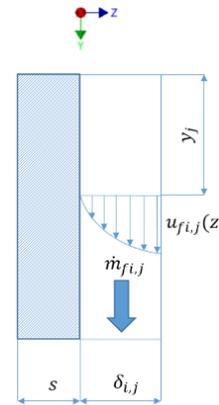
$$u_f(z) = \frac{\rho_f - \rho_g}{\mu_l} g \left[ \delta z - \frac{1}{2} z^2 \right] + \frac{\tau_g}{\mu_l} z. \quad (9)$$

Where range for z coordinate is  $0 \leq z \leq \delta_{j,i}$ ,  $\rho_f$  – condensate film density,  $\rho_g$  – mixture density and  $\tau_g$  is shear stress at interface between mixture and film. Local mean velocity in condensate film is obtained by integration over local film thickness

$$\bar{u}_f(x, y) = \bar{u}_{f,j,i} = \frac{1}{\delta_{j,i}} \int_0^{\delta_{j,i}} u_f(x, y, z) dz. \quad (10)$$

Local mass flow of condensate film is given by

$$\dot{m}_{f,j,i} = \rho_f \bar{u}_{f,j,i} \delta_{j,i} \Delta x. \quad (11)$$



**Fig. 7.** Viscosity effect in condensate film

These equations are used to solve local film thickness -  $\delta_{i,j}$ , that is necessary for solving heat transfer through film.

### 2.6.2 Film Temperature solver

Local temperature of film on tubes and fins is very important parameter for heat transfer and condensation and it is determined from heat balance between humid air, film and fin/tube surface. [3] [6]

Heat balance between air, film and surface can be written as

$$k'(T_F - T_{fw}) = \dot{m}_f'' \Delta h_{23} + \alpha_{fg} E_T (T_G - T_F). \quad (12)$$

Increment of film mass flow

$$\dot{m}_f'' = \rho_g \beta_g \ln \left( \frac{1 - \tilde{y}_v \{T_F\}}{1 - y_v} \right), \quad (13)$$

$$\phi_T = \frac{\rho_g \beta_g c_{pv}}{\alpha_{fg}} \ln \left( \frac{1 - \tilde{y}_v \{T_F\}}{1 - y_v} \right). \quad (14)$$

Combining equations above we obtain

$$k'(T_F - T_{fw}) = \alpha_{fg} \phi_T \left( \frac{\Delta h_{23}}{c_{pv}} + \frac{T_G - T_F}{1 - \exp(-\phi_T)} \right). \quad (15)$$

Equation (15) is solved iteratively

$$\begin{aligned} &\rightarrow T_F^{i+1} = \\ T_c + \frac{\alpha_{fg} \phi_T}{k'} \left( \frac{\Delta h_{23}}{c_{pv}} + \frac{T_G - T_F^i}{1 - \exp(-\phi_T)} \right). \end{aligned} \quad (16)$$

From equation (16), mean film temperature is solved after couple iterations. This equation is also used as a part of solution stability monitoring – the first indication of solution divergence is unstable film temperature, if this fact is detected, simulation is terminated.

**2.6.3 Fin surface temperature solver**

After considering simplifications, problem of fin surface temperature can be then solved separately (based on informations from air and refrigerant side) as 2D heat conduction problem in rectangular domain. To ensure unique solution, boundary conditions are required – refrigerant temperatures at fin/tube intersections are used. As a volumetric heat loads, local heat flow from gas (or film) into fin is used. [7] [2]

Heat flow (flux) vector is

$$\mathbf{q} = \begin{bmatrix} q_x \\ q_y \end{bmatrix}. \tag{17}$$

Fourier’s law in matrix form can be written as

$$\mathbf{q} = -\mathbf{D}\nabla T. \tag{18}$$

Linear shape functions ( $N$ ) are chosen for interpolation, weak formulation is established as

$$\begin{aligned} \iint_{\Omega} \mathbf{B}^T \mathbf{s} \mathbf{D} \mathbf{B} d\Omega \mathbf{a} = \\ - \oint_{\partial\Omega_q} \mathbf{N}^T t q_b d\partial\Omega - \oint_{\partial\Omega_T} \mathbf{N}^T s q_n d\partial\Omega \\ + \iint_{\Omega} \mathbf{N} s Q d\Omega. \end{aligned} \tag{19}$$

Where

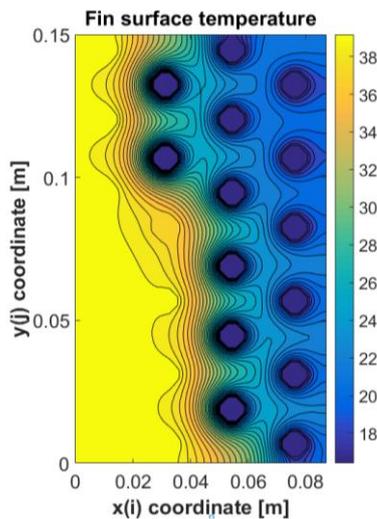
$$\mathbf{B} = \nabla \mathbf{N}, \tag{20}$$

$$\mathbf{a} = [T_i]. \tag{21}$$

Equations from weak formulation can be then rewritten as a simple matrix equation

$$\mathbf{K} \mathbf{a} = \mathbf{f}_b + \mathbf{f}_l = \mathbf{f}. \tag{22}$$

This method is then applied on heat exchanger geometry for arbitrary inlet conditions, resulting surface temperature is shown in **Fig. 8**.



**Fig. 8.** Fin surface temperature

Because plate/fin surface temperature is solved every iteration, code optimization is required to decrease overall computation time even for fine meshes. Temperature in mesh nodes can be solved from equation (22) by

$$\mathbf{a} = \mathbf{K}^{-1} \mathbf{f} \tag{23}$$

Matrix inverse operation (equation (23)) is very demanding and imprecise for large matrices. The dimension of stiffness matrix for a coarse mesh is typically 7300x7300, which takes 22 seconds on average (4 core CPU, 3,3GHz). This is not suitable for simulations that usually take 150 iterations to converge. Matrix inverse can be replaced by *mldivide* operation implemented in MATLAB. Another improvement is using algorithms for sparse matrices (CRS form – compressed row storage) and LU factorization.

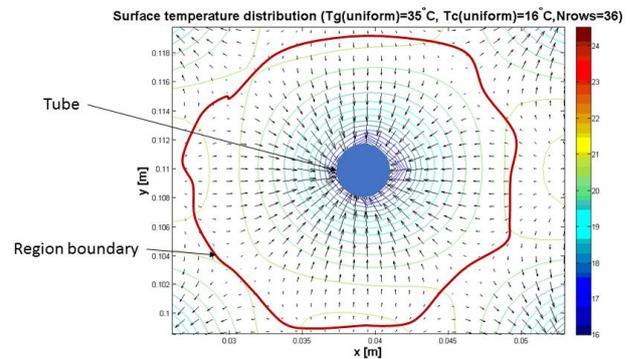
$$[\mathbf{L}_K, \mathbf{U}_K] = lu(\mathbf{K}_{sparse}) \tag{24}$$

Equation (23) is then replaced by 2 equations

$$\tilde{\mathbf{a}}_{temp} = \mathbf{L}_K \setminus \mathbf{f} \tag{25}$$

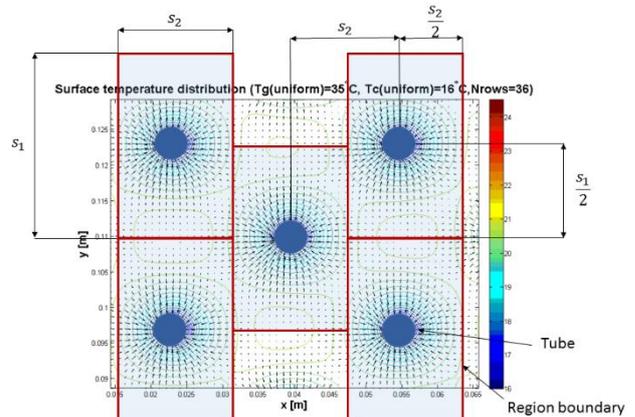
$$\mathbf{a} = \mathbf{U}_K \setminus \tilde{\mathbf{a}}_{temp}. \tag{26}$$

Plate/fin surface also needs to be divided into sub-regions (areas), from where all the heat flow coming from the air is assigned to specific tube cross section. This allows evaluating enthalpy/temperature increment of tube pass. In general, these sub-regions can be traced by finding locations, where local heat flux inside the fin is 0 or very small. A boundary of these sub-regions are very complicated (**Fig. 9**); so an additional simplification is required.



**Fig. 9.** Complex fin sub-regions

Simplified sub-regions of fin are shown in **Fig. 10**.



**Fig. 10.** Simplified fin sub-regions



experimental study of evaporator performance have been carried out. Requirements and properties of the mathematical model are introduced. The MATLAB software is selected as a model development environment, because of its user-friendly environment, the relatively simple implementation of matrix operations and the available optimization tools for model validation and correction. The model itself is composed of multiple sub-procedures that solve various heat and mass transfer problems – fin surface temperature, condensate film temperature, local humid air parameters and conditions for condensation as well as the refrigerant temperature distribution along the tube bundle. The differential equations for heat and mass balance used in the model are solved with explicit first order Euler method. As a possible improvement of MATLAB model, additional code optimization and generalization can be mentioned: the current computation time is approximately 2 second per iteration for reasonable mesh density. Instead of using a finite difference method for the air temperature distribution and a finite element method for surface temperature distribution, it is preferable to use a 2D finite volume method, which may lead to further decrease in the computation time. The next step is validation of the model. It will be divided into multiple stages, to test all necessary components of MATLAB model and maximize the precision of results. As a continuation of this work, the model of the condenser should be created. It can be simply derived from evaporator model, by excluding condensation procedures and other simplifications, so further time reduction of computation time is expected. To complete the whole heat pump circuit, simplified model of a compressor and reduction valve should be also developed, to be able to simulate circuit as a unit, without measurement of particular parameters, like refrigerant temperature in condenser outlet and evaporator inlet, etc.

We gratefully acknowledge the support from Centre of 3D volumetric velocimetry (COLA), reg. no. CZ.2.16/3.1.00/21569, supported by operational programme Prague Competitiveness.

<b>Nomenclature</b>		
<b>Used symbols</b>		
<i>Symbol</i>	<i>Unit</i>	<i>Name - description</i>
$\mathbf{q}$	$W \cdot m^{-2}$	heat flux vector
$\mathbf{D}$	$W \cdot m^{-1} \cdot K^{-1}$	thermal conductivity matrix
$\nabla T$	$K \cdot m^{-1}$	temperature gradient
$k$	$W \cdot m^{-2} \cdot K^{-1}$	overall heat transfer coefficient
$p$	Pa	pressure
$p_0$	Pa	barometric pressure
$p_v$	Pa	water vapor saturation pressure
$T$	°C, K	temperature
$\beta$	$m \cdot s^{-1}$	mass transfer coefficient
$s$	mm, m	fin/plate thickness
$\delta$	mm, m	condensate film thickness
$h$	$J \cdot kg^{-1}$	enthalpy
$q_{gw}$	$W \cdot m^{-2}$	heat flux from air to fin/film
$q_b, q_n$	$W \cdot m^{-2}$	boundary heat flows
$Q$	$W \cdot m^{-3}$	internal heat supply

$\Omega$	-	area of elements
$\Omega_{q,T}$	-	elements boundary with heat or temperature condition
$\mu$	Pa · s	dynamic viscosity
$\nu$	$m^2 \cdot s^{-1}$	kinematic viscosity
$\tau_g$	Pa	shear stress at air-film interface
$R$	$J \cdot mol^{-1} \cdot K^{-1}$	gas constant
$u$	$m \cdot s^{-1}$	velocity
$\dot{m}$	$kg \cdot s^{-1}$	mass flow rate
$\dot{m}''$	$kg \cdot m^{-2} \cdot s^{-1}$	mass flux
$I_{230}$	$J \cdot kg^{-1}$	latent heat of evaporation
$\Delta h_{23}$	$J \cdot kg^{-1}$	enthalpy of evaporation
$\alpha$	$W \cdot m^{-2} \cdot s^{-1}$	heat transfer coefficient
$N$	1	number of elements in x direction
$M$	1	number of elements in y direction
$Err_{Surf}$	%	error of tube section area
$q_b, q_n$	$W \cdot m^{-2}$	boundary heat flows
$Telem$	1	number of elements in tube cross-section
$\lambda$	$W \cdot m^{-1} \cdot K^{-1}$	heat/thermal conductivity
$y$	1	mole fraction
$\tilde{y}$	1	mole fraction at saturation
$c_p$	$J \cdot kg^{-1} \cdot K^{-1}$	specific heat/heat capacity at constant pressure

#### Subscripts

$a$	air
$w$	wall (fin or tube surface)
$v$	water vapor
$f/film$	condensate film
$c$	coolant
$a$	air

#### References

1. Reversing valve cooling. [Ecorenovator](#). © EcoRenovator (2012)
2. M. Borovička, Student's conf. STC, 7 (2016)
3. Li Jun-De, S. Mohammad, G. Thorpe. *IJHMT*. **54**, 4078 (2011).
4. J. Šesták, F. Rieger, *Přenos hybnosti, tepla a hmoty*. (CTU, Prague, 2005)
5. Verein Deutscher Ingenieure. *VDI-Wärmeatlas*. (7<sup>th</sup> ed., VDI-Verlag, 1994)
6. F. P. Incropera, *Fundamentals of heat and mass transfer*. (7<sup>th</sup> ed. John Wiley& Sons, Inc, USA, 2011)
7. AALBORG UNIVERSITET. *Finite Element Method – 2D heat conduction* (2015)
8. J. Pavlů, *Vývoj výpočetního modelu a metodiky pro výpočet kondenzátorů s minikanálky* (Czech technical university in Brno, Faculty of Mechanical Engineering, Brno, 2012)
9. J. Adamec, *Modelování přestupu tepla a hmoty ve filmové výplni* (CTU, Faculty of Mechanical Engineering, Department of Fluid Dynamics and Thermodynamics, Prague, 2014)