Mathematical model of compact type evaporator

Martin Borovička¹, *, Tomáš Hyhlík¹

¹CTU 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.

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.
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)

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.

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.
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

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]

\[ \frac{du_f}{dz} = \frac{\rho_f - \rho_g}{\mu_1} g(\delta - z) + \frac{\tau_g}{\mu_1}. \]  

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

\[ u_f(z) = \frac{\rho_f - \rho_g}{\mu_1} g \left[ \delta z - \frac{1}{2}z^2 \right] + \frac{\tau_g}{\mu_1} z. \]  

Where range for z coordinate is \(0 \leq z \leq \delta_{i,j}\), \(\rho_f\) – 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

\[ \bar{u}_f(x,y) = \frac{1}{\delta_{i,j}} \int_{0}^{\delta_{i,j}} u_f(x,y,z) dz. \]  

Local mass flow of condensate film is given by

\[ m_{f,i,j} = \rho_f \bar{u}_f(x,y) \delta_{i,j} \Delta x. \]  

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

\[ k'(T_F - T_{fw}) = \frac{\alpha_f}{\ell} \Delta h_{23} + \alpha_f E_f(T_a - T_F). \]  

Increment of film mass flow

\[ m''_f = \rho_f \beta_f \ln \left( \frac{1 - \bar{y}_f(T_F)}{1 - \bar{y}_f(T_a)} \right). \]

Combining equations above we obtain

\[ k'(T_F - T_{fw}) = \alpha_f \phi_f \frac{T_h}{c_{pv}} \ln \left( \frac{1 - \bar{y}_f(T_F)}{1 - \bar{y}_f(T_a)} \right). \]

Equation (15) is solved iteratively

\[ T_F^{i+1} = \frac{T_c + \alpha_f \phi_f \frac{T_h}{c_{pv}} \left( \Delta h_{23} + \frac{T_g - T_F}{1 - \exp(-\phi_f)} \right)}{k'}. \]
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

\[ q = \begin{bmatrix} q_x \n q_y \end{bmatrix}. \] (17)

Fourier’s law in matrix form can be written as

\[ q = -D \nabla T. \] (18)

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

\[ \int_{\Omega} B^T s DB \n abla a = \]

\[ - \int_{\partial\Omega_4} N^T q_b d\partial \Omega + \int_{\partial\Omega_4} N^T q_a d\partial \Omega \]

\[ + \int_{\Omega} N s Q d\Omega. \] (19)

Where

\[ B = \nabla N, \]

\[ a = [T_i]. \] (20) (21)

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

\[ Ka = f_b + f_i = f. \] (22)

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

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 solved from equation (22) by

\[ a = K^{-1} f \] (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 \texttt{mldivide} operation implemented in MATLAB. Another improvement is using algorithms for sparse matrices (CRS form – compressed row storage) and LU factorization.

\[ [L_K, U_K] = lu(K_{sparse}) \] (24)

Equation (23) is then replaced by 2 equations

\[ \bar{a}_{temp} = L_K \bar{f} \]

\[ a = U_K \bar{a}_{temp}. \] (25) (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.
This simplification also brings new problem – for a case with big temperature gradients between 2 neighboring tubes, some of the heat that in reality “flows” into tube 1 is within this simplification associated with tube 2. This causes an error in heat balance (difference of enthalpy on air side is not equal to enthalpy on the refrigerant side).

To prevent this situation, Redistribution Correction Solver is added. RCS eliminates this error by transferring additional heat from 1 sub-region to the other, based on estimation from heat flux integral on its boundary.

2.6.4 Superposition of heat and mass transfer

Theoretical heat transfer coefficient on fin and tube is known, but effect of tube presence on heat transfer coefficient on fins (and fin presence to heat transfer coefficient on tubes), needs to be included. If we assume that evaporator is only composed of tubes, total heat duty without condensation (mass transfer) can be written symbolically as [2] [8]

$$Q_{\text{tubes}} = \tilde{a}_{\text{tube}} \cdot A \cdot \left(T_g - T_f\right),$$

(27)

where $\tilde{a}_{\text{tube}}$ is heat transfer coefficient on tubes. Similarly, if evaporator (heat exchanger) is made only from fins, total heat duty is

$$Q_{\text{fins}} = \tilde{a}_{\text{plate}} \cdot A \cdot \left(T_g - T_f\right),$$

(28)

where $\tilde{a}_{\text{plate}}$ is heat transfer coefficient on fins/plates. Superposing fins and tubes, total heat duty without condensation (mass transfer) can be symbolically:

$$Q_{\text{tot}} = \left(R_p \cdot \tilde{a}_{\text{plate}} + R_t \cdot \tilde{a}_{\text{tube}}\right) \cdot A \cdot \left(T_g - T_f\right).$$

(29)

Where superposition (effectiveness) coefficients $R_p$, $R_t$ are derived from experimental data by optimization – fitting the model results to results from experiment. These coefficients describe influence of temperature and velocity field of one case to the other. Similar hypothesis can be made with condensate mass flow, so another 2 effectiveness coefficients are introduced. Local heat and mass transfer coefficients are obtained from correlations for the case of flow in small slot (as a limit of concentric annular duct) and flow over staggered tube bundle. These coefficients are then used to solve overall heat transfer coefficient according to Fig. 11.

2.6.5 POD and RBF method

Another incorporating of data transfer process is done by incorporating the discrete version of POD (Proper Orthogonal Decomposition) method - Method of Snapshots. This method allows to simplify a large amount of data into a smaller package and also to reconstruct solution for given (directly unsolved) inlet parameters very quickly. Input for POD method is set of direct MATLAB solutions for different parameters – inlet conditions.

All fields from solution are then decomposed into average values and fluctuation, which is then decomposed into modes and amplitudes. Reconstructed solution for given inlet conditions can be written symbolically as [9]

$$\text{OUTPUT}_{\text{param}_k} = \text{OUTPUT}_{\text{mean}} + \sum_{j=1}^{n} \text{MOD}_j \cdot \text{AMPLITUDE}_{\text{param}_j}$$

(30)

where $n$ is number of modes. Fields $\text{OUTPUT}_{\text{mean}}$ and $\text{MOD}$ are constant, but $\text{AMPLITUDE}$ is a function of inlet conditions that needs to be constructed based on direct solutions with Radial Basis Functions (RBF) in Gaussian form. Overall approximation function can be interpreted as RBF network.

$$h(x) = b + \sum_{k=1}^{N} w_k e^{-\gamma \|x - x_k\|^2}.$$  

(31)

Where $x$ is arbitrary points in n-D parameter space and $x_k$ is a point for which direct simulation is performed. Coefficients $b$ and $w_k$ are trained with data from direct solutions by MATLAB optimization method $fminsearch$. Target (fitness) function is defined as

$$\Delta_{\text{tot}} = \sum_{k=1}^{N} \Delta(x_k),$$

(32)

where sample error function is evaluated as

$$\Delta(x_k) = (h(x_k) - y_k)^2.$$  

(33)

Total approximation error function can be then written as

$$\Delta_{\text{tot}} = \sum_{k=1}^{N} \Delta(x_k) = \sum_{k=1}^{N} \left(b + \sum_{l=1}^{n} w_l e^{-\gamma \|x_k - x_l\|^2} - y_k\right)^2.$$  

(34)

Necessary number of modes required to “precise” approximation is determined from quantity called level of correlation and it is defined as

$$L_n = \frac{\sum_{j=1}^{n} \lambda_j}{\sum_{j=1}^{n} 1}; \quad n = 1, 2, ..., N$$

(35)

Where $N$ is a number of direct simulations and $n$ is a number of modes, choose so $L_n > 0.99$. Implementation of POD and RBF method into the mathematical model is covered more deeply in [2].

Conclusion

The main subject of this paper is the heat and mass transfer modeling inside heat exchangers (evaporator) used in air dehumidification applications. Theoretical and
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 seconds per iteration for reasonable mesh density. Instead of using a finite difference method for the air temperature distribution along the tube bundle, the precision of results. As a continuation of this work, the necessary components of MATLAB model and maximize further time reduction of computation time is expected. To complete the whole heat pump circuit, simplified condensation procedures and other simplifications, so that 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 also be 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.

| Symbol | Unit         | Name - description                              |
|--------|--------------|--------------------------------------------------|
| $q$    | W $\cdot$ m$^{-2}$ | heat flux vector                                |
| $D$    | W $\cdot$ m$^{-2}$ $\cdot$ K$^{-1}$ | thermal conductivity matrix                      |
| $\beta$ | K $\cdot$ m$^{-2}$ $\cdot$ K$^{-1}$ | temperature gradient                            |
| $p_0$  | Pa           | barometric pressure                              |
| $p_f$  | Pa           | water vapor saturation pressure                  |
| $\theta$ | °C, K     | temperature                                     |
| $\beta$ | m $\cdot$ s$^{-1}$ | mass transfer coefficient                       |
| $s$    | mm, m       | condensate film thickness                        |
| $h_b$  | J $\cdot$ kg$^{-1}$ | enthalpy                                        |
| $q_{gw}$ | W $\cdot$ m$^{-2}$ | heat flux from air to fin/film                   |
| $q_p$, $q_n$ | W $\cdot$ m$^{-2}$ | boundary heat flows                              |
| $Q$    | W $\cdot$ m$^{-3}$ | internal heat supply                             |

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. (7th ed., VDI-Verlag, 1994)
6. F. P. Incropera, Fundamentals of heat and mass transfer. (7th 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 minikandálky (Czech technical university in Brno, Faculty of Mechanical Engineering, Brno, 2012)
9. J. Adamec, Modelování přestupu tepla a hmoty ve filmově výplně (CTU, Faculty of Mechanical Engineering, Department of Fluid Dynamics and Thermodynamics, Prague, 2014)