Numerical Modelling of Heat Transfer in Convector’s Pipes by ABAQUS

Zh D Kolev¹ and S Y Kadirova²

¹Department of Heat, Hydraulics and Environmental Engineering, University of Ruse, Bulgaria
²Department of Electronics, University of Ruse, Bulgaria

Email: zkolev@uni-ruse.bg

Abstract. The paper presents method for implementation of numerical modelling of heat transfer process between the water flow and the internal surface of a "water-air" convector’s pipes. The convector is a heat consumer in a reversible heat pump installation. To implement the CFD simulations, which are part of the numerical modelling of the processes, the ABAQUS software has been used. The main features and results from the simulations have been analysed. The method includes experimental investigations aiming determination of the boundary conditions for CFD simulation. The distribution of temperature, pressure and velocity in the fluid flowing through the pipe have been investigated. The results of the investigation of the temperature distribution present reduced heat transfer intensity in pipe’s sections, characterized by higher local hydraulic resistances.

1. Introduction

One of the most common heat transfer processes in the heat exchangers is the heat transfer process between flowing in a pipe fluid and the pipe’s internal surface. These processes are characterized by complexity of influencing factors, which lead to difficulties in their numerical modeling. On the other hand, the investigation of the processes of heat transfer making accurate prediction for design of the heat exchangers [1, 2]. The numerical simulations verify the accuracy and reliability of experimental results [3].

In order to accomplish numerical modeling of such processes and to obtain results regarding the distribution of certain parameters (temperature, pressure, velocity, etc.), experimental investigation is needed to define the input data for CFD simulation [1, 2, 3, 4, 5].

When solving numerical heat transfer problems, use of correct boundary conditions is compulsory. The problem and its numerical modeling will lead to establishment of adequate model with high precision [6].

The three-dimensional CFD simulation of heat transfer processes allow obtaining of information about parameters, which can’t be observed experimentally [5, 7, 8, 9].

One of the softwares suitable for computer simulations of heat transfer processes is ABAQUS (ABAQUS FEA engineering software). It is based on finite elements method (FEM) and can be applied for different engineering fields [3, 4, 9, 10].

The “water-water” heat pump installations are characterized with water temperature variation in the heat consumers. Therefore, the boundary conditions are defined for specific operation parameters of the installation [11].
In many studies heat transfer processes have been considered generally. Average values of the heat transfer parameters have been calculated and no detailed analysis has been implemented in respect of certain zones characterized by reduced heat transfer intensity. In this study a quantitative investigation of zones characterized by reduced intensity of the heat transfer process has been implemented.

The aim of the paper is to define heat transfer processes parameters in the convector’s pipes, based on CFD modelling.

2. Methods for determination of the input data for CFD simulation

The experimental reversible “water-water” heat pump installation can operate in two regimes - "heating" and "cooling". The installation consists of internal and external circles. The heat consumer is a "water-air" convector, located in the internal circle of the installation [11]. This research focuses on investigation of the heat transfer processes by forced convection in the convector’s pipes.

The CFD modelling of the processes can be divided into a few parts. The first one refers to preparing the geometry [1, 3, 5, 9]. Three dimensional model (finite element 3D model) for the water flow has been developed in the current research. CFD is fulfilled by splitting a fluid domain into small cells for creating the mesh [9].

Schematic diagram of the considered simulation model is presented in Figure 1.

Figure 1. Schematic diagram of the simulation model.

- Measurement of the temperature in the center of the pipe’s inlet cross-section (inlet bulk temperature) \( T_{f,1} \), °C;
- Measurement of the temperature in the center of the pipe’s outlet cross-section (outlet bulk temperature) \( T_{f,2} \), °C;
- Calculation of the heat flow \( Q \), W (Eq.1);

\[
Q = \dot{m} \cdot c_p \cdot \Delta T_f, W
\]

where: \( \dot{m} \) is the mass flow rate, kg/s; \( c_p \) - mass specific heat capacity (at constant pressure), J/(kg.K);
\( \Delta T_f \) – fluid temperature difference between pipe’s inlet and outlet cross-sections, K.
- Calculation of the input average fluid velocity \( \bar{U}_{input} \), m/s, based on the measured water volumetric flow rate, measured temperature \( T_{f,1} \), the number of the pipes, and calculated cross-sectional area of the pipe.
- Measurement of the outlet fluid manometric pressure \( p_{m, output} \), Pa.

3. Results and discussion

3.1. Initial conditions for CFD simulation

The "Tet" shape of the computing cells has been set. In order to receive good regular mesh, an approximate size of 0.005 m of the computing cells is set. The computational domain is discretized by using 69293 cells (calculation model’s elements). The result for computing nodes is 18384.
3.2. Input data for CFD simulation

Numerical modelling of two heat transfer processes in the convector’s pipes has been implemented. The input data is determined by the developed methods. Dimensions of fluid flow are assumed to: \( d = 8 \) mm, and \( l = 4087 \) mm. These are the geometric dimensions of the internal surface of the convector’s pipes [11]. The input data of the simulated processes are shown in Table 1.

Table 1. Input data of the simulated heat transfer processes.

| Operation regime of the installation | \( \bar{U}_{\text{input}}, \text{m/s} \) | \( p_{\text{in/out}}, \text{Pa} \) | \( T_{f,1}, ^\circ\text{C} \) | \( T_{f,2}, ^\circ\text{C} \) | \( q, \text{W/m}^2 \) |
|-------------------------------------|----------------|----------------|----------------|----------------|----------------|
| “heating“                           | 1.472          | 150000         | 50.8           | 42.0           | 26220.2        |
| “cooling“                           | 2.536          | 150000         | 7.5            | 12.4           | 25569.2        |

3.3. Validation of CFD model

Validation aims to verify the accuracy of CFD model [1, 3]. In study [1] the numerical results on heat transfer and pressure drop are validated using theoretical correlations. In [2] the used parameter for validation of CFD model is the average temperature of the investigated process.

In the present study, the validation is performed by comparing the simulation and experimental values of the average heat convection coefficient, between the fluid, flowing in the convector’s pipes, and their internal heat exchange surface. For this purpose, the data of the „cooling“ regime of the installation is used (Table 1).

The simulation value of the heat convection coefficient is calculated using the average cross-section temperature difference, regarding to the pipe’s length. For this purpose results of the temperature field were taken from the CFD post.

The experimental value of the heat convection coefficient is calculated according to a criterion equation, using the measured inlet and outlet pipe’s temperatures.

The calculated error obtained from the validation of the model is 4.92 %.

3.4. Simulation results for temperature field

The simulation results, regarding the data in Table 1 are presented in Figure 2.

Figure 2. Temperature field in longitudinal cross-section.

The variation of the fluid temperature depends on the heat flow direction. At “heating” regime the temperature decreases in direction of flow, forasmuch as the heat flow direction is to the pipe’s internal surface. In “cooling” regime is seen the opposite, respectively.

3.5. Investigation of zones with reduced heat transfer intensity

Figure 3 presents zones with relatively decreased amount of temperature of the external cylindrical surface of the fluid, which is the pipe’s internal heat exchange surface, in "heating" regime. Low temperatures indicate relatively low intensity of the heat transfer. This is an indicator of high temperature difference in the pipe’s cross-section. There is a necessity of quantitative investigation of the influence of hydraulic resistances on the heat transfer intensity.
Similarly, the zones with visible reduced intensity of the heat transfer process in "cooling" regime are presented in Figure 4. The location of these zones is analogous to the location of the zones, formed during the operation of the installation in "heating" regime. Relatively high surface temperatures are observed, which are indicators of a relatively large temperature difference in the pipe’s cross-sections and for reduced heat transfer intensity.

![Zones with reduced heat transfer intensity in “heating” regime](image1)

**Figure 3.** Zones with reduced heat transfer intensity in “heating” regime

The investigated cross-sections (Fig. 3 and Fig. 4) are presented in Table 2.

![Zones with reduced heat transfer intensity in “cooling” regime](image2)

**Figure 4.** Zones with reduced heat transfer intensity in “cooling” regime.

| Cross-section | A    | B    | C    | D    |
|---------------|------|------|------|------|
| Zone          | I    | II   | III  | IV   |
| Distance from the pipe’s inlet, m | 0.021267 | 0.023827 | 0.405538 | 0.403126 |

Indicator of the reduced heat transfer intensity in the region of the respective cross-section is the ratio $\Delta T_H / \Delta T_N$. The parameters $\Delta T_H$ and $\Delta T_N$ are respectively relatively high temperature difference and normal temperature difference, specific for relevant the cross-section.

The nodes for temperature differences $\Delta T_H$ and $\Delta T_N$ are defined in “heating” regime (Figure 5). A node located close to the center of the cross-section’s center has been chosen for determination of the temperature differences. For each investigated cross-section it has been established in advance that the difference between the temperatures of the nodes, located near the center of the cross-section, is negligible. For determination of $\Delta T_H$, a node located on the external cylindrical surface of the fluid, in the region with relatively low temperature, has been selected. For $\Delta T_N$, the selected node from the external surface is in a region with a normal surface temperature for the cross-section.
Similarly, the CFD results for the cross-sections in “cooling” regime have been plotted in Figure 6. The node selected from the external fluid surface, used for calculation of $\Delta T_H$, is with relatively high temperature.
Table 3 contains results obtained from the CFD post, for the investigated in "heating" and "cooling" regimes.

**Table 3. Results for "heating" and "cooling" regimes.**

| Regime  | Cross-section | $\Delta T_H$, °C | $\Delta T_N$, °C | $\Delta T_H/\Delta T_N$ [-] |
|---------|---------------|-----------------|-----------------|-----------------|
| "heating" | “A” | 8.752 | 3.415 | 2.563 |
|         | “B” | 8.676 | 3.919 | 2.144 |
|         | “C” | 7.587 | 3.133 | 2.423 |
|         | “D” | 7.342 | 3.500 | 2.098 |
| "cooling" | “A” | 9.781 | 3.634 | 2.691 |
|         | “B” | 9.259 | 3.959 | 2.338 |
|         | “C” | 8.181 | 3.207 | 2.551 |
|         | “D” | 7.729 | 3.531 | 2.189 |

The ratio $\Delta T_H/\Delta T_N$ is with higher values for the cross-sections, closer to the knees’ centers, comparing to those closer to the linear pipe’s sections ("A" with "B"; and "C" with "D", respectively). Therefore, the local hydraulic resistances in the cross-sections, closer to the knees’ centers, have higher impact on the heat transfer intensity.

3.6. **Investigation of the influence of the fluid velocity on the heat transfer intensity**

The investigation has been realized at various heat transfer process parameters, obtained by control of the average fluid velocity $\bar{U}_{input}$ in the pipe. Based on the uniform quality change of the temperature at the different fluid velocity studied, the selected cross-section is "D". The study is in “heating” regime.

The parameters of the heat transfer process and the results obtained from CFD modelling, for the temperature differences $\Delta T_H$ and $\Delta T_N$, are presented in Table 4.

**Table 4. Results for the temperature differences.**

| $\bar{U}_{input}$, m/s | $T_{f,1}$, °C | $T_{f,2}$, °C | $q$, W/m² | $\Delta T_H$, °C | $\Delta T_N$, °C |
|------------------------|--------------|--------------|--------|----------------|----------------|
| 1.389                  | 50.8         | 42.4         | 23624.9 | 6.857          | 3.289          |
| 1.592                  | 50.8         | 43.4         | 23836.2 | 6.327          | 2.982          |
| 1.860                  | 50.8         | 44.5         | 23709.9 | 5.658          | 2.618          |
| 2.267                  | 50.8         | 45.6         | 23838.6 | 4.852          | 2.229          |
| 2.470                  | 50.8         | 45.8         | 24974.3 | 4.768          | 2.179          |
Figure 7 presents the graphical relation between the ratio of temperature differences and the average fluid velocity in the pipe.

![Graphical representation of the function](image)

Figure 7. Graphical representation of the function $\frac{\Delta T_H}{\Delta T_N} = f(\overline{U}_{\text{input}})$. 

From the investigation can be concluded that at higher velocity values, the curve is less sloping, i.e. the difference in ratio values decrease. Hence the increase in fluid velocity results in decrease in the rate of decrease of the heat transfer intensity.

4. Conclusion

This investigation based on CFD modeling of the heat transfer process in the convector’s pipes, obtains results for the temperature distribution. This information is used for determination of the heat convection coefficient in the pipes of the heat exchanger.

Results for quantitative temperature variation between defined computing nodes have been obtained.

The accuracy of the results obtained from the CFD simulation depends on the set number of computing cells (the grid density). The calculated model’s error is 4.92%. The built finite element computing mesh provides the necessary quality of results simultaneously with not very long duration of the simulations.

From the three-dimensional numerical analysis is found that the local hydraulic resistances in the pipe influence on the intensity of the heat transfer process. The increase of the fluid velocity results in a decrease in the heat transfer intensity in the investigated cross-sections’ regions of the pipe’s knees. The variation of the fluid velocity in the range of 1.39 to 2.47 m/s corresponds to increase of the temperature differences ratio from 2.084 to 2.188 (by 4.75 %).

References

[1] Cebrucean D and Cebrucean V 2017 Numerical study of a shell-and-tube heat exchanger for heating rich monoethanolamine using hot flue gases – Part II. Tube-side heat transfer enhancement, Revista Termotehnica. XXII (2) pp 37-41

[2] Osley W, Droegeemueller P and Ellerby P 2013 CFD investigation of heat transfer and flow patterns in tube side laminar flow and the potential for enhancement, Chemical Engineering Transactions 35 pp 997–1002

[3] Oon C, Togun H, Kazi S, Badarudin A, Zubir M and Sadeghinezhad E 2012 Numerical simulation of heat transfer to separation air flow in an annular pipe, International Communications in Heat and Mass Transfer 39 pp 1176–1180

[4] Abdelkader K, Youcef K and Sabiha T 2016 Numerical simulation of free vibrations of multilayer composite beams, Proceedings of 2nd Conference on Advances in Mechanical Engineering ICAME2016, Istanbul, Turkey, p. 105–109
[5] Yordanov K, Mechkarova T, Stoyanova A and Zlateva P 2018 Determination of the temperature of cathode unit of indirect plasma burner through a computer simulation model, Proceedings of the Second International Scientific Conference “Intelligent Information Technologies for Industry” (IITI’17), pp 403-409

[6] Yordanov K, Zlateva P, Hadzhidimov I and Stoyanova A 2018 Testing and clearing the high temperature module error from 0 to 1250°C for measurement with 16 K-type thermocouples, Proceedings of the 20th International Symposium on Electrical Apparatus and Technologies (SIELA), pp 480-483

[7] Han-Taw C, Yu-Jie C, Chein-Shan L and Jiang-Ren C 2017 Numerical and experimental study of natural convection heat transfer characteristics for vertical annular finned tube heat exchanger, *International Journal of Heat and Mass Transfer* **109** pp 378–392

[8] Majewski K and Grądziel S 2016 CFD simulations of heat transfer in internally helically ribbed tubes, *Chemical and Process Engineering* **37** (2) pp 251-260

[9] Saternus Z and Piekarska W 2017 Numerical analysis of thermomechanical phenomena in laser welded pipe-to-flat, *Procedia Engineering* **17** pp 196-203

[10] Steinheimer R and Engel B 2014 Thermal influences during rotary draw bending of tubes from stainless steel, *Procedia Engineering* **81** pp 2165–2170

[11] Kolev Zh, Kadirova S and Nenov T 2017 Research of reversible heat pump installation for greenhouse heating, *INMATEH - Agricultural Engineering* **2** pp 77-84