X CONGRESSO NACIONAL DE ENGENHARIA MECÂNICA 2018
20 A 24 DE MAIO DE 2018 SALVADOR – BA – BRASIL

EXPERIMENTAL AND COMPUTATIONAL SIMULATION OF A CONCENTRIC TUBES HEAT EXCHANGER: THERMAL EFFECTIVENESS AND NUMERIC MODEL MICRO FINS ADDITION ANALYSIS

Breno Gusmão Barbosa, brenogusm@gmail.com[1]
Estêvão Lannes Tolentino, estevaotolentino@hotmail.com[1]
Julio Ziviani Pochmann, julioziviani@hotmail.com[1]
Frederico Romagnoli Silveira Lima, fredericolima@cefetmg.br[1]

[1]Centro Federal de Educação Tecnológica de Minas Gerais, Department of Mechanical Engineering, Av. Amazonas, nº 5.253, Nova Suiça, 30.421-169, Belo Horizonte, Brazil.

Abstract. *A concentric tubes heat exchanger belonging to the thermal fluid dynamics laboratory of the Centro Federal de Educação Tecnológica de Minas Gerais was analyzed. The heat exchanger was experimentally studied in a laboratorial test bench and a numerical finite element model was created using the COMSOL Multiphysics software. Input parameters as hot and cold side inlet water temperatures and flow rates were varied in order to gather output data of the heat exchanger as thermal effectiveness. The experimental and numeric results were compared to validate the created model. The model definition of material, finite element mesh assembled and the appropriate boundary conditions were set to faithfully represent the physicals phenomena of heat transfer and fluid mechanics involved. The thermal effectiveness variation with addition of annular micro fins on the external surface of the inner tube of the heat exchanger was simulated and analyzed. Through a parametric study, the raise of 10% in thermal effectiveness could be achieved with the micro fins assembly.*

Keywords: *concentric tubes heat exchanger, COMSOL, thermal effectiveness, micro fins.*

1 INTRODUCTION

The heat exchangers are devices widely used in industry and are responsible for enhancing the heat transfer between two fluids at different temperatures (Oliveira, 2012). The heat transfer between two fluids is a process used in many applications such as power generation, steam purification and petroleum refining. When the transfer of heat to accomplish the phase change is necessary, heat exchangers are designated as condensers, when it is desired condensing a fluid or evaporators when it is desired evaporation of the fluid.

The configuration of a concentric tube heat exchanger is defined as a tube inside another. In the inner tube flows a fluid, usually the heated one and in the annular space of the configuration, usually circulates the cooling fluid. The relative movement of fluids can be parallel or countercurrent. In parallel motion, the hot and cold fluids enter in the same edge, flow in the same direction and leave the heat exchanger in the same end. In the counter flow arrangement, the fluids enter into opposite edges, flow in opposite directions and exit through opposite ends.

The amount of thermal energy exchanged by the fluids, hot and cold, depend on parameters such as inlet temperatures and velocities, heat transfer area and the flow path.

One way to improve thermal efficiency of a concentric tube heat exchanger is by increasing the contact area between the cooling fluid and the surface of the inner tube with the addition of fins. However, the new configuration also generate pressure drop and stagnation points, which reduces the heat exchange. Therefore, a study of the behavior of the thermal effectiveness is needed to see how it is improved.

In this paper, a numerical model was developed and validated using the experimental data. After validating the model, the effectiveness of the heat exchanger with the addition of micro fins on the inner tube was analyzed. A concentric tubes heat exchanger numerical model was developed and validated by using experimental data. The heat exchanger thermal behavior was analyzed (Naphon, 2006). A validated model was obtained and the effectiveness enhances with the addition of micro fins on the inner tube was simulated and analyzed (Naphon, 2011).

2 GOVERNING EQUATIONS

According to Bergman et al. (2011), the effectiveness of a heat exchanger (ε) is the ratio between the real heat transfer rate and the maximum possible heat transfer rate, and can be calculated by Equation (1), which assesses the decreasing in temperature difference at the output. A heat exchanger in which the fluids leaves it at the same temperature corresponds to an exchanger with an effectiveness of 100%.

$$\varepsilon=\frac{q_{real}}{q_{max}}=1-\frac{T_{out,hot}-T_{out,cold}}{T_{in,hot}-T_{in,cold}},\quad 0\leq\varepsilon\leq1 \tag{1}$$

The governing equations used by COMSOL are shown in equations (2), (3) and (4):

$$\rho(\mathbf{u}\cdot\nabla)\mathbf{u}=\nabla\cdot\left[-\rho\mathbf{I}+\mu(\nabla\mathbf{u}+(\nabla\mathbf{u})^{T})-\frac{2}{3}\mu(\nabla\cdot\mathbf{u})\mathbf{I}\right]+\mathbf{F} \tag{2}$$

$$\nabla\cdot(\rho\mathbf{u})=0 \tag{3}$$

$$\rho C_p\mathbf{u}\cdot\nabla T=\nabla\cdot(k\nabla T)+Q \tag{4}$$

The model is governed by continuity equations, energy conservation and Navier-Stokes equations.

3 METHODS

With the objective of obtain the computational heat exchanger model, empirical laboratorial scientific experiments for data gathering were realized. For the development of the proposed study, steps have been set for the construction of a reliable numerical model of the concentric tubes heat exchanger. The results obtained are capable of realistic interpretations. The concentric tube heat exchanger model created was validated by comparing the variable parameters obtained experimentally with data derived from the computer simulation.

It was attempted to get a mesh that reached a convergence with an error smaller than 1%, what would allow an approximate interpretation of the real model. For this, a study of the tools that COMSOL Multiphysics provides was conducted for the creation of the mesh.

After obtaining a reliable model, it was studied the effectiveness of the heat exchanger. To calculate the effectiveness, it was considered the temperature variation at the entrance and exit of the heat exchanger. After the study of the effectiveness of the model created, small changes in the model geometry were made. Such modifications are the introduction of micro fins in the external surface of the inner tube in order to create new configurations that potentiate the thermal exchange, thereby increasing the effectiveness of the device.

3.1 Numeric model

To develop the model in COMSOL Multiphysics, it was settled simplifications for the heat exchanger of the study. These simplifications have been studied to see if they do not alter the expected results. After creation, calibration and validation of the model, micro fins were inserted into the stainless steel inner tube to study the effectiveness gain in the new configuration.

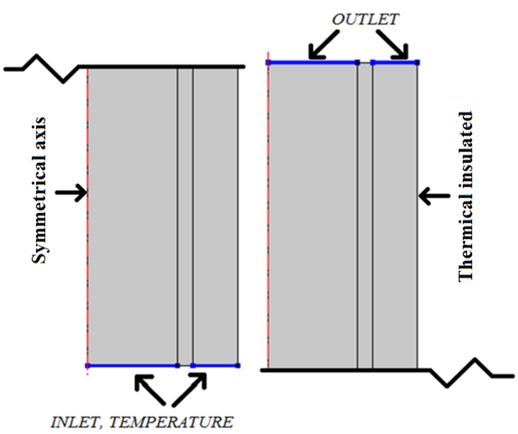

Figure 1. Model boundary conditions.

It was developed an axis-symmetric 2D model of the heat exchanger. In this model, it was considered the heat transfer that occurs between the fluids through the stainless steel inner tube. In the axis-symmetric 2D model, a rectangle that is rotated 360 degrees about its axis of symmetry was constructed, to represent the cylindrical tube.

In the definition of the boundary conditions of the created model, the COMSOL Multiphysics software functions Temperature, Inlet and Outlet were used, as shown in Fig. 1.

For the mesh refinement were used the functions Size, Boundary Layers and Free Triangular. To selected the mesh, the output outlet temperature of the fluids in several mesh sizes were compared. After testing the mesh sizes, for a practical model study, without spent much time to simulate all the experimental data and with a less than 1% error, it was chosen the Coarse mesh size for the simulation of the model built.

The materials used in COMSOL were water, and AISI 304 stainless steel with density of 7900 kg/m³, thermal conductivity of 14.9 W/m²K and specific heat at constant pressure of 477 J/kgK according to Bergman et al. (2011).

The concentric tubes heat exchanger has 520 mm as effective length of heat transfer between fluids. The annular region has an internal diameter of 12 mm and external diameter of 20 mm. The inner tube made of stainless steel have a thickness of 1 mm and the outer tube is made of acrylic plastic with a wall thickness of 10 mm and it has a thermal conductivity of 0.18 W/mK, it acts as an isolate material. The heat rate dissipated through the acrylic wall is irrelevant in the experiment, so the acrylic tube was not considered.

The area of the inner tube through which the hot fluid flows is 1.13E-4 m², and the area of the annular region through which the cold fluid flows is 1.60E-4 m².

The natural convection occurs in connection pipes, where the hot and cold fluids flow without exchanging heat, was calculated. The values founded didn’t show a relevant natural convection in the studied heat exchanger, reaching approximately 8.50 W/m²K for the connection tube with hot fluid in an adopted average temperature of 326.05K. The effective heat transfer between the fluids occurs in the in stainless steel pipe.

In the experimental study, there were small variations of parameters that should be constants, i.e. the input temperature. However, as these variations were not relevant, average values were considered in the COMSOL Multiphysics model. With design and materials chose, and with the established considerations, it was possible to obtain the concentric tubes heat exchanger model in the COMSOL Multiphysics.

3.2 Experimental Data

The collection of experimental data was realized in the test bench of the laboratory of thermal fluid dynamic of CEFET-MG. The bench is from TECQUIPMENT and the model is TD360. The heat exchanger concentric tubes simulated in the study, of the same manufacturer, is TD360A.

The test bench works with a supply of hot and cold water to the heat exchanger pipes, with a flow rate controlled by a system with needle valve and turbine flow meter. The hot fluid input temperature is regulated by a lung tank and the cold fluid input temperature is the same of the hydraulic network of the environment. In Fig. 2 the test bench is showed.

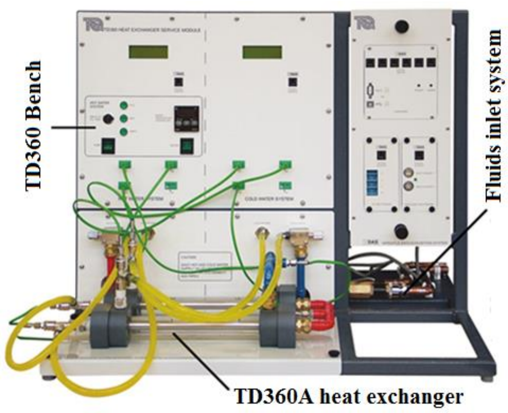

Figure 2. Test bench.

To obtain the experimental data, the test bench has sensors and transmitters that are connected to the heat exchanger, allowing collecting the temperature in certain points. Each sensor is connected to the bench by a green cable.

The data are collected by a proper test bench software and can be recorded in equal time intervals, allowing a steady state study of the heat exchanger.

The test bench can be configured in different forms, allowing that the hot and cold fluids flow in parallel direction or in counter-current direction. The experimental data obtained allowed to realize studies about the heat exchanger effectiveness.

The studied heat exchanger has concentric tubes, which allows the heat transfer between the fluids by the wall of the inner tube. The hot fluid flows by the inner tube and the cold fluid flows by the annular region created by the configuration. The heat exchanger constructive form divides it in two parts that are connected by connection tubes. The blue tubes are used with the cold fluid and the red tubes are used with the hot fluid.

The concentric tubes heat exchanger, Fig. 3, has three temperature measurement points for each fluid, connected to the test bench. The first one in the input, the second in the middle of the path and the last one in the output.

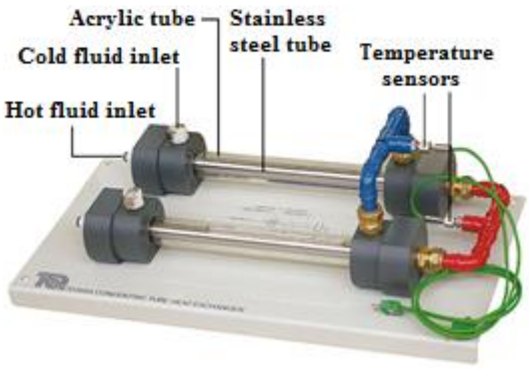

Figure 3. Concentric tubes heat exchanger.

The TD360A heat exchanger effectiveness is not influenced by the flow direction. Therefore, in the study only parallel flux was adopted, Tab. 1. In the first experiment, the cold fluid flow was changed. In the second experiment the hot fluid input temperature was changed. The experiment was realized at 296.85 K (environment temperature).

Table 1. Parallel flow experiments.

Experiment	1			2		
Set	A	B	C	D	E	F
$V_{in,h}$ [m/s]	0.44	0.44	0.44	0.44	0.44	0.44
$V_{in,c}$ [m/s]	0.26	0.21	0.11	0.44	0.44	0.44
$T_{in,h}$ [K]	332.87	333.44	333.55	309.58	318.40	328.01
$T_{out,h}$ [K]	327.71	328.6	330.25	307.85	315.29	323.82
$T_{in,c}$ [K]	296.65	296.72	296.65	295.65	295.45	295.65
$T_{out,c}$ [K]	302.11	303.15	305.53	297.95	299.45	301.24
ε [%]	29	31	33	29	31	30

Where $V_{in,h}$ is the hot fluid velocity inlet, $V_{in,c}$ [m/s] the cold fluid velocity inlet, $T_{in,h}$ and $T_{out,h}$ the temperature of the hot fluid in the inlet and outlet respectively, $T_{in,c}$ and $T_{out,c}$ the temperature of the cold fluid in the inlet and outlet respectively, and ε the heat exchanger effectiveness.

4 RESULTS AND DISCUSSIONS

The configurations sets of Tab. 1 were simulated. All sets were done with the COMSOL Multiphysics tool called Parametric Sweep, it allows the variation of some chosen parameters for each experiment, without needing to process each configuration individually. For the simulations the input speed was varied and for the experiment one, and the hot fluid input temperature was changed for experiment 2. Using the tool Parametric Sweep, the results were obtained for each experiment in a total simulation time of 2898 s, according to Tab. 2.

Table 2. Output temperatures obtained with the model compared with Table 1.

Simulation	1			2		
Set	A	B	C	D	E	F
$T_{out,h}$ [K] (COMSOL)	329.25	329.45	330.41	307.83	315.48	323.80
$T_{out,h}$ [K] (EXP.)	327.71	328.60	330.25	307.85	315.29	323.82
$T_{out,c}$ [K] (COMSOL)	301.74	302.93	305.20	297.55	298.53	299.60
$T_{out,c}$ [K] (EXP.)	302.11	303.15	305.53	297.95	299.45	301.24
ε [%]	26	29	32	24	24	24

The model showed high degree of approximation with the experimental values. Regarding to heat exchanger effectiveness, the simulation number one was closer to experimental values than the simulation number two.

The considerations in the numerical model that were done helped to improve the data processing, once only the relevant parameters were computed. The effect of the cold fluid input velocity could be simulated and the temperatures profiles obtained in the heat exchanger output could be observed in Fig. 4.

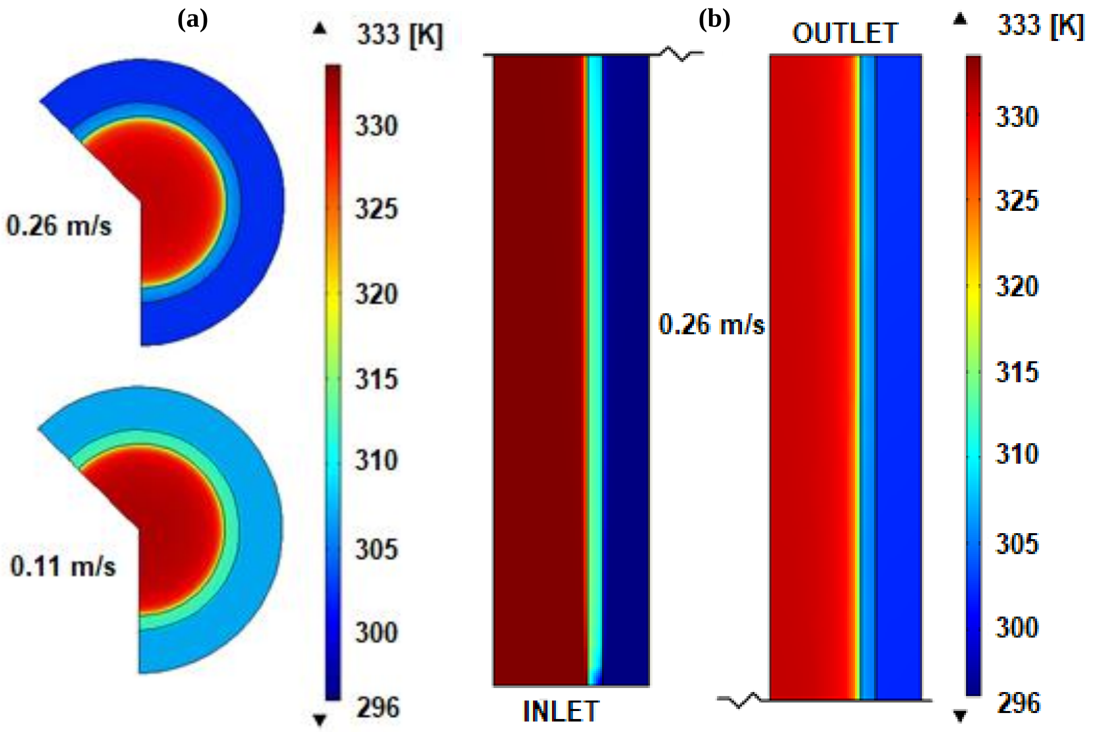

Figure 4. Experiment 1 thermal behaviors of parallel flow. (a) cross view outlet temperature distribution for sets A and C; (b) axial view inlet and outlet temperature distribution for set A.

4.1 Introducing micro fins

With the addition of annular micro fins for a numerical simulation, the physical phenomena of the model with and without the fins could be studied and compared. The analyzed behaviors showed the heat transfer augmentation expected but it came with the effects of fluid mechanical head loss, and stagnancy regions, as consequence, where there was less convective heat transfer. A configuration study was realized in order to improve the input parameters and reduce the negative impacts of those loss effects (Andrade and Zaparoli, 1999).

A parametric study, similar as the one Mohanty et al. (2014) did with twisted tapes, was made in order to find a configuration where the passive augmentation technique stablished a better relation between the head losses and stagnancy areas in order to do not diminish the effectiveness. As the modeled heat exchanger is commonly used for academicals purposes and has small scale dimensions, the friction factor of the fins addition that generates the head losses in the fluid showed to be minimal, where the stagnancy regions were the main impact factor in the thermal effectiveness. What is not the case for industrial heat exchangers of large scale, where the additional pump energy required could have a greater impact.

A satisfactory numerical model was obtained for the study by the addition of annular micro fins on the outer surface of the stainless steel inner tube. To increase the effectiveness of the concentric tubes heat exchanger, micro fins were added to the model and six different configurations of height, width and spacing between fins were studied. The mesh was also considered Coarse because COMSOL can refine the mesh in accordance with the geometric parameters.

The configuration set A from Tab. 2 was studied. Using the Parametric Sweep function, it was possible to study the height (H), width (W) and spacing between fins (S) with just a single command to start the processing of the program. The output temperature values obtained for each configuration is shown in Table 3. The simulation time was 3603 s.

Table 3. Finned model. Length of 1 mm.

H [mm]	1	1	1	1	2	2	2	2
W [mm]	0.25	0.25	0.5	0.5	0.25	0.25	0.5	0.5
S [mm]	5	10	5	10	5	10	5	10
$T_{out,h}$ [K]	327.72	327.63	327.65	327.29	327.81	327.4	327.53	327.45
$T_{out,c}$ [K]	303.14	303.54	302.96	303.43	302.79	303.4	302.95	303.25

ε [%]	34	35	34	36	33	35	34	35

The addition of micro fins in the model reduced the temperature difference between the hot and cold fluids in the outlet. Without the use of fins, the temperature difference was 27.5 °C and with the addition of fins, the temperature difference became about 24 °C, decreasing 3.5 °C.

The advantage of using a computer program such as COMSOL is precisely when a problem becomes too complex to manually resolving. The gain of thermal effectiveness was nearly 10% with the addition of micro fins. The effectiveness for no fin in experiment one set A model was approximately 26%, and the effectiveness for finned model reached 36%.

In Fig. 5 it can be seen the velocity behavior of the cold and hot fluids during the passage through the heat exchanger for the fin with configuration 2 mm in height, 0.25 mm in thickness and 10 mm spacing.

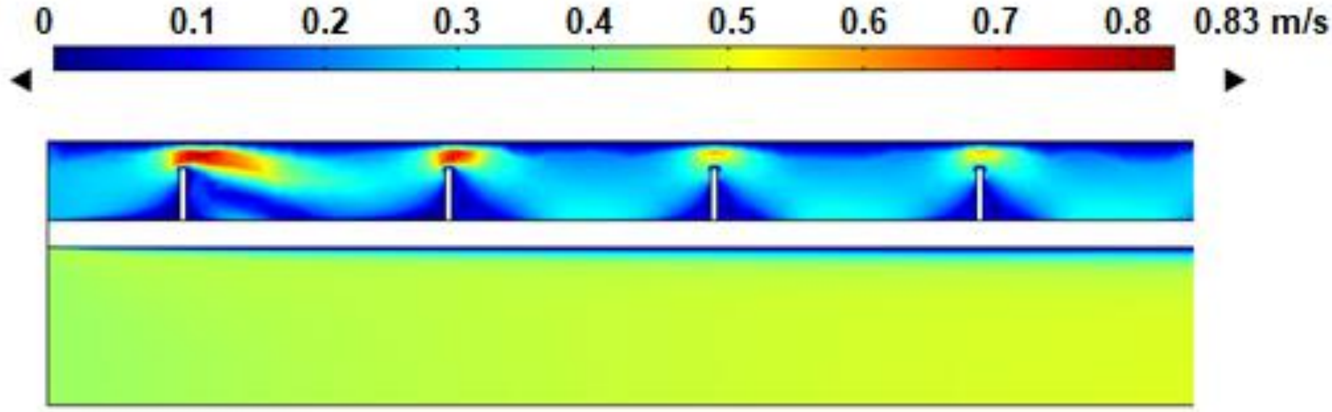

Figure 5. Velocity profile in the first 4 fins in finned heat exchanger.

To improve the visibility of the thermal effects on the micro fins, another case was simulated with fins having 0.5 mm in height, 0.1 mm in thickness and 5 mm of spacing. The temperature profile can be seen in Fig. 6.

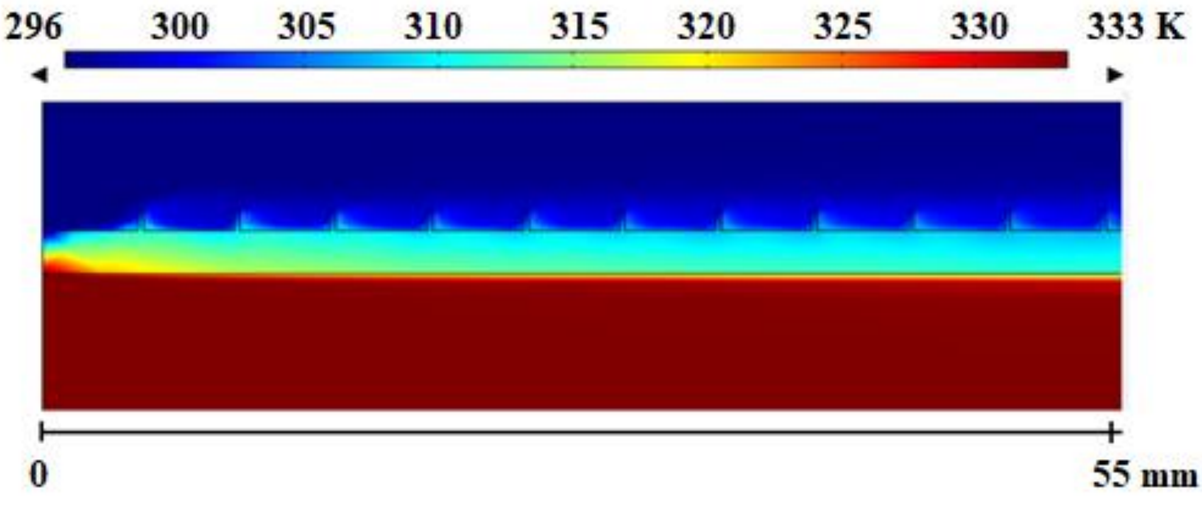

Figure 6. Temperature profile at the inlet of the new configuration of micro fins.

In Fig. 6, the stagnant fluid region behind the fins, considering the fluid flow direction from left to the right, results in a decrease of thermal exchange between fluids, because of the temperature difference of the fluids reduction near the stainless steel wall.

5 CONCLUSIONS

The study showed satisfactory results about the numerical model of the existing heat exchanger in thermal fluid dynamics lab of CEFET-MG.

COMSOL Multiphysics showed to be accurate in the simulation of a real concentric tube heat exchanger for the finite element mesh type, size and geometrical considerations of the computational model used.

The numerical simulation used showed to be a good tool for pre-feasibility parametric studies, it can help in the creation of suitable new experimental concentric tubes heat exchangers. With the validated computer model, it was possible to estimate the concentric tubes heat exchanger effectiveness gain with the addition of annular micro fins. The best effectiveness gain was reached within a parametric study of selected settings values of 1 mm in height, 0.25 mm in thickness and 5 mm of spacing between the fins. The addition of micro fins produced a thermal effectiveness gain approximately of 10 %, what decrease the cold and hot water outlet temperatures difference by 3.5 °C, compared with the same boundary conditions without the micro fins.

The 2D axis-symmetric model, which considered only the area of heat transfer between fluids, showed good approximation to the data obtained experimentally. The remarks made in order to simplify the model and optimize the speed and processing time were highly relevant for the study.

The determining factors for the construction of the models were geometries, types of mesh and boundary conditions.

With the numerical simulation, most suitable real models can be created for tests in the study of new settings. With the computer model, it was possible to quantize the gain effectiveness with the addition of the anular micro fins.

6 ACKNOWLEDGEMENTS

We would like to thank Centro Federal de Educação Tecnológica de Minas Gerais (CEFETMG) for the use of the termal fluid dynamic laboratory and for all the technical material and resources necessary to conduct the experiments carried out in this work.

7 REFERENCES

de Andrade, C. R., Zaparoli, E. L. 1999. "Otimização de parâmetros de transferência de calor no escoamento em uma região anular aletada". In: *XV Congresso Brasileiro de Engenharia Mecânica*, Águas de Lindóia, São Paulo.

Bergman, T. L., Lavine, A. S., Incropera F. P., Dewitt D. P. 2011. *Fundamentals of Heat and Mass Transfer*. John Wiley & Sons, New Jersey, 7th edition.

Mohanty R. L., Bashyam, S., Das, D., 2014. "Numerical analysis of double pipe heat exchanger using heat transfer augmentation techniques", *International Journal of Plastics Technology*, Vol. 18(3), p. 337-348.

Naphon, P., 2006. "Second law analysis on the heat transfer of the horizontal concentric tube heat exchanger", *International Communications in Heat and Mass Transfer*, Vol. 33, p. 1029-1041.

Naphon, P., 2011. "Study on the exergy loss of the horizontal concentric micro-fin tube heat exchanger", *International Communications in Heat and Mass Transfer*, Vol. 33, p. 229-235.

Oliveira, A. C. G. 2012. *Modelação térmica e hidrodinâmica de escoamentos em permutadores de calor*. M.Sc. Thesis, Universidade do Minho, Portugal.

8 RESPONSIBILITY NOTICE

The authors are the only responsible for the printed material included in this paper.