

ENCIT-2018-0609 MATHEMATICAL MODEL AND CFD SIMULATION OF A DOUBLE TUBE HEAT EXCHANGER

Daniel Alberto Flórez Morales
Luz Elena Peñaranda Chenche
Abdul Orlando Cardenas Gomez
Enio Pedone Bandarra Filho

Federal University of Uberlandia. Mechanical school. Uberlandia (MG), Brazil. Av Joao Naves de Avila, 2121.
Esdanielflorez@gmail.com
Ing.elenap@gmail.com
Orlandocardenas1589@gmail.com
bandarra@ufu.br

***Abstract.** In this article, a mathematical model and numerical simulation of the turbulent heat transfer process in a double tube heat exchanger has been carried out. Experimental test were performed to evaluate the heat transferred and the pressure drop of the device in parallel flow and counter flow. The model accept empirical correlations proposed by different authors. The finite volume method with the standard k-e turbulence model is used to solve the equations of continuity, momentum, energy and turbulence in three-dimensional domain. They were measured and compared with the corresponding data of the correlations and numerical study. The data was met with an error of 6% and an error.*

***Keywords:** Heat transfer coefficient, double tube heat exchanger, heat transfer, method volume finite*

1. INTRODUCTION

The double tube heat exchangers are mainly used in the industry, fulfilling the function of transferring energy to a fluid. In applications, the objective could be to recover or reject heat, pasteurize, distill or control a fluid in a process. The increase in the amount of heat generated by industrial processes requires more efficient energy systems, of reduced dimensions and with a lower operational cost. Establishing the mathematical model and the numerical simulation that can predict the thermal behavior helps to understand the functioning of these devices getting to evaluate the heat transfer and pressure drops of the process and leading to future investigations in the improvement of the efficiency and the field of diagnosis and failure prevention of these devices.

2. MATHEMATICAL MODEL

In the mathematical model, the mathematical formulas involved in the double tube heat exchanger devices are described step by step, thus elaborating a mathematical model that allows analyzing and quantifying the energy interactions that occur inside it. For the resolution of it, it was subdivided into control volumes along the x direction and the output temperatures, heat transfer coefficient, effectiveness and pressure drops for the input data were calculated Input temperatures and mass flows.

2.1 Physical model

The concentric tubes have a length of 5.64 m, are made of copper with a thermal conductivity ($k = 200\text{W} / \text{m} \cdot \text{K}$). The inner tube has an outer and inner diameter of 0.0425m and 0.0373m respectively. The shell has an inside diameter of 0.073m. The outer side of the shell is isolated. The required properties of the working fluids are defined using functions obtained from the tables available in the literature (Cengel, 2011).

2.2 Considerations of the model

To raise the mathematical model, the following considerations were taken into account:

- The double tube heat exchanger operates in a stationary state.
- The outer surface of the exchanger is perfectly insulated.
- Temperatures and entry speeds are measurable.

- Changes in kinetic and potential energy are negligible.

2.3 Governing equations

The concentric tubes have a length of 5.64 m, are made of copper with a thermal conductivity ($k = 200\text{W} / \text{m} \cdot \text{K}$). The inner tube has an outer and inner diameter of 0.0425m and 0.0373m respectively. The shell has an inside diameter of 0.073m. The outer side of the shell is isolated. The required properties of the working fluids are defined using functions obtained from the tables available in the literature (Cengel, 2011).

The conservation equations for permanent regime are applied. The subscripts "h" and "c" refer to the value of the hot fluid and cold fluid. The subscripts "in" and "out" refer to the value in the input and output respectively.

Mass conservation equation

The mass balance for each of the control volumes is expressed as:

$$\sum \dot{m}_{in} = \sum \dot{m}_{out} \quad (1)$$

Where \dot{m} is mass flow.

Energy conservation equation

The energy balance for a control volume in the form of heat flux in a permanent regime applying the first thermodynamic law is expressed as:

$$\sum \dot{E}_{in} = \sum \dot{E}_{out} \quad (2)$$

Thus,

$$Q = \dot{m}_h C_{p,h} (T_{h,inlet} - T_{h,outlet}) \quad (3)$$

$$Q = \dot{m}_c C_{p,c} (T_{c,outlet} - T_{c,inlet}) \quad (4)$$

Where C_p is the specific heat and T is the temperature.

Newton's Cooling Law

Heat transfer can also be expressed according to Newton's cooling law as:

$$Q = UA_S \Delta T_m \quad (5)$$

Where U is the global heat transfer coefficient, A_S is the heat transfer area and ΔT_m is the average temperature difference.

2.4 Effectiveness-NTU method

For this study the exit temperatures are unknown since our interest lies in the calculation of them for the validation of the mathematical model and the numerical simulation. Therefore, the NTU- ϵ method is used (Kays and London, 1955).

This method based on a dimensionless parameter called effectiveness of heat transfer ϵ defined as:

$$\epsilon = \frac{Q_{real}}{Q_{max}} \quad (6)$$

Where " Q_{real} " is the current heat flow obtained by making the energy balance and Q_{max} is the maximum possible heat flow. The effectiveness " ϵ " for a double concentric tube heat exchanger for each flow arrangement is expressed:

Parallel

$$\epsilon = \frac{1 - \exp(-NTU(1+c))}{1+c} \quad (7)$$

Counterflow

$$\varepsilon = \frac{1 - \exp(-NTU(1-c))}{1 - c \exp(-NTU(1-c))} \quad (8)$$

Where NTU is the number of transfer units and "c" is the ratio of capacities and are expressed as:

$$NTU = \frac{UA_S}{c_{min}} \quad (9)$$

$$c = \frac{c_{min}}{c_{max}} \quad (10)$$

$$Q = \varepsilon \dot{Q}_{max} \quad (11)$$

In this way we can calculate the temperatures at the exit for each of the fluids such as:

$$T_{h,out} = T_{h,in} + \left(\frac{Q}{C_h}\right) \quad (12)$$

$$T_{c,out} = T_{c,in} + \left(\frac{Q}{C_c}\right) \quad (13)$$

Where C_h y C_c is the heat capacity of each flow.

Calculation of pressure drop

For a fully developed flow the pressure drop in the inner and ring flow can be calculated as follows:

$$\Delta P_L = f \frac{L}{D} \frac{\rho V_{avg}^2}{2} \quad (14)$$

Where f is the friction factor of Darcy, L is the length of the tube, D is the hydraulic diameter, ρ is the density of the fluid at the inlet temperature and V_{avg} is the average velocity.

3. SIMULATION

To solve the governing equations continuity, momentum and energy in three dimensions, the ANSYS FLUENT 17 software was used, considering border conditions equivalent to those generated in the mathematical model and in the experimental procedure. The following summarizes the assumptions, simplifications and boundary conditions used.

3.1 Assumptions for simulation

- Stable state: It is assumed that any point of the flow will not change its state over time.
- Incompressible flow: Because the fluid is a liquid at temperatures lower than the saturation, the change in its specific volume is not considered with the variation of the pressure.
- Single phase flow: Phase change is not considered and heat transfer is studied in the case of a pure substance.
- Turbulent flow: The standard model to be used corresponds to the classic model of two equations k- ε
- Gravitational field: The effects of gravity on the flow patterns are considered.

3.2 Geometry

The geometry is the starting point of the process, in this step you define the spatial domain you want to study, in this case you take the dimensions of the heat exchanger at a scale of 1:1 and proceed to make the model in the Design module ANSYS Modeler as shown in the following figure.

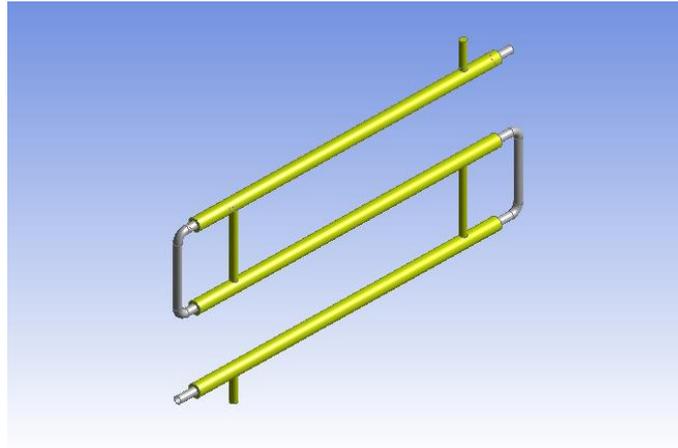


Figure 1. 3D model of the heat exchanger

3.3 Characteristics of the mesh used

The mesh process is fundamental to develop the simulation, in this part the domain was subdivided to be solved by the processor, tetrahedral elements were chosen for the discretization of the domain.

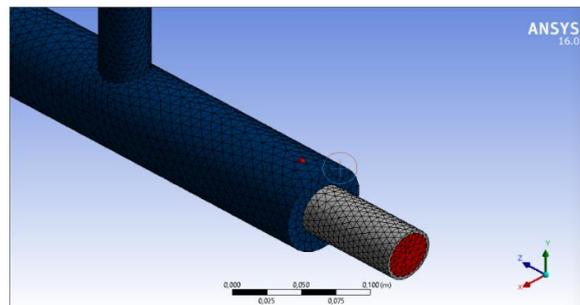


Figure 2. Tetrahedral mesh

Both the shape of the elements and the number of elements is of great importance for the solution, because the greater number of elements in theory would have better results, however, the computational cost rises, this is undesired by the resources with the what is counted. This is why a mesh independence analysis is carried out in which different numbers of elements are available for the same boundary conditions. The results are shown in the following table and figure:

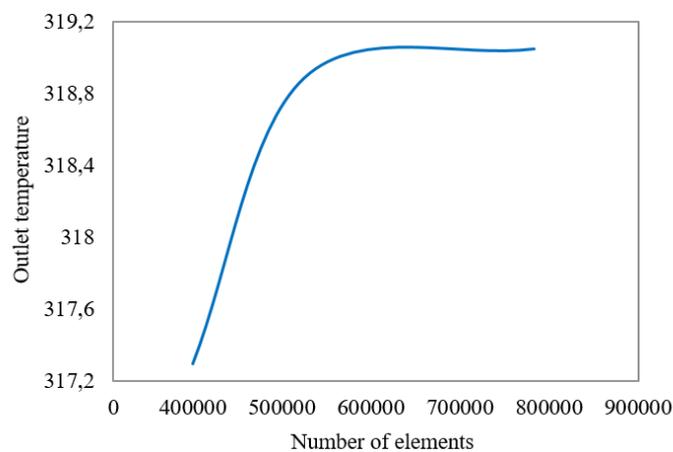


Figure 3. Mesh independence analysis

Table 1. Number of elements

Number of elements	T_cal (K)
400000	317,3
525697	318,89
783851	319,05

As it can be observed in the results for the hot water output, it shows variations between the meshes of 525697 and 783851 lower than 0.16 ° C, however, in calculation time it was 2 hours and 50 minutes, therefore, choose the mesh of 525697 elements to perform the simulations. The main characteristics are summarized in the following table:

Table 2. Characteristics of the mesh used

Parameter	Value
Method of assembly of the mesh	Cut Cell
Relevance center	Fine
Smoothing	Medio
Aspect ratio	4.63
Minimal orthogonal quality	0.53
Minimal orthogonal symmetry	0.3
Number of nodes	408000
Number of elements	525697

3.4 Boundary conditions

The boundary conditions for the simulation are defined as shown in the following table:

Table 3. Boundary conditions

Boundary conditions	
Name	Condition
Inlet_Hot	mass-flow-inlet
Inlet_Cold	mass-flow-inlet
Outlet_Hot	pressure outlet
Outlet_Cold	pressure outlet
Cold interface_Tube	coupled Wall
Hot interface_Tube	coupled Wall
External walls	heat flux 0

4. RESULTS

After having established the mathematical model and the numerical simulation in the present study, the detailed results are presented: temperature distribution, heat transfer coefficient, global performance and pressure drops for the double tube heat exchanger.

4.1 Temperature distribution

4.1.1 Temperature distribution in mathematical model of EES

The volume of control of each fluid was subdivided into 30 control volumes with a $\Delta x = 0.188\text{m}$, the temperatures in each of the points of interest were calculated by solving the equations previously presented along the x direction in the software of EES, achieving results that coincide with what is observed in the literature (Cengel, 2015).

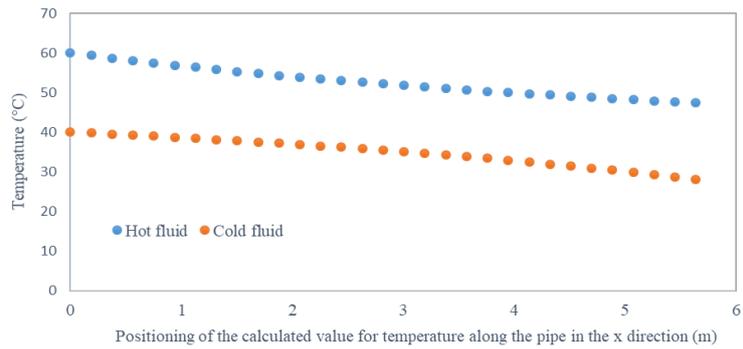


Figure 4. Counterflow temperature distribution

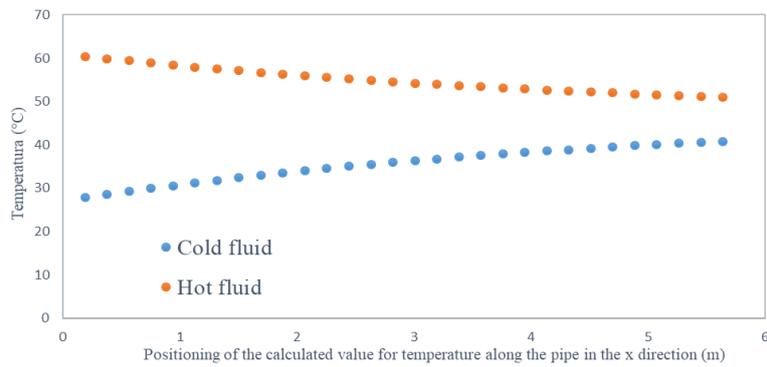


Figure 5. Parallel flow temperature distribution

4.1.2 Temperature distribution in the ANSYS FLUENT

The results in ANSYS FLUENT are presented for all the simulations as shown in Fig.6.

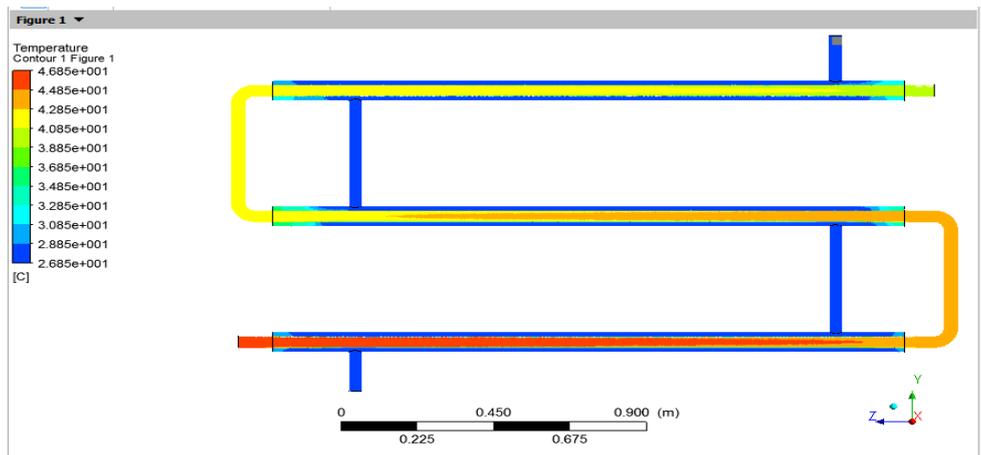


Figure 6. Temperature distribution in the ANSYS FLUENT

4.2 Coefficients of heat transfer by convection

The variation of the coefficient of heat transfer by convection of the tube inside is presented for different mass flows \dot{m}_h and temperatures of the hot fluid $T_{h,inlet}$. The input parameters for the cold fluid are set a $\dot{m}_c = 0,1065 \text{ kg/s}$, $T_{c,inlet} = 28 \text{ }^\circ\text{C}$

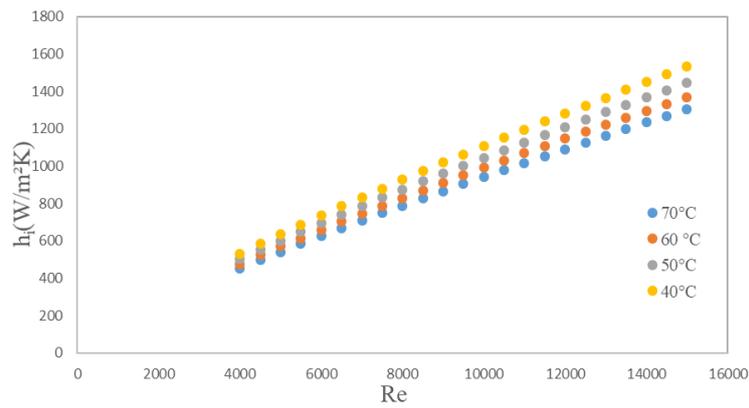


Figure 7. Variation of the heat transfer coefficient by internal convection for different mass flows and different hot fluid temperatures.

Where, Re is the Reynolds number.

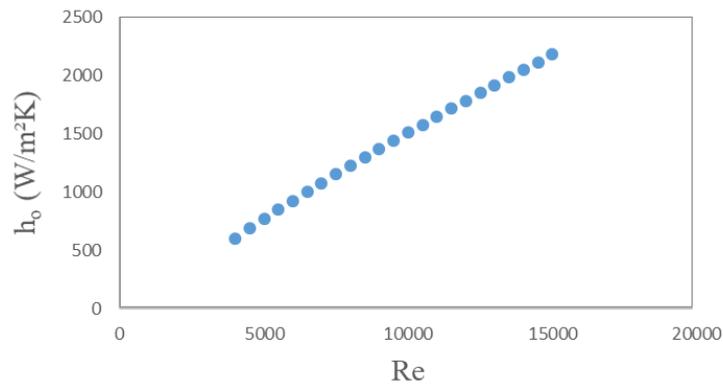


Figure 8. Variation of the heat transfer coefficient by annular convection for different mass flows at the feed temperature.

4.3 Pressure drops

The results of pressure drops for each of the flows of the double tube heat exchanger are presented:

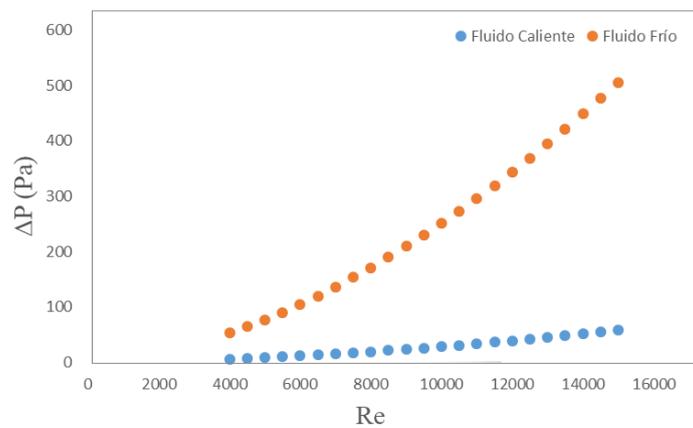


Figure 9. Pressure drops in the heat exchanger

5. COMPARISON RESULTS

Next, the results obtained by the mathematical model and the CFD simulation are presented, as well as the error percentage of them compared to the results of the experimental tests for each of the established parameters.

Table 4. Parallel results

PARALLEL RESULTS													
Inlet temperature		Flow		Mathematical model		ANSYS FLUENT		Experimental tests		%Error mathematical model		%Error ANSYS	
T _{h, inlet} (°C)	T _{c, inlet} (°C)	V _h (m ³ /s)	V _c (m ³ /s)	T _{h, outlet} (°C)	T _{c, outlet} (°C)	T _{h, outlet} (°C)	T _{c, outlet} (°C)	T _{h, outlet} (°C)	T _{c, outlet} (°C)	Hot	Cold	Hot	Cold
47	27	1,82e ⁻⁴	7,72e ⁻⁴	37,3	29,2	36,7	29,6	36	29	3,61	0,68	1,94	2,06
51	27	1,05e ⁻⁴	5,01e ⁻⁴	38,61	31,27	37,86	31	37	30	4,35	4,23	2,32	3,33
55	27	1,23e ⁻⁴	7,9e ⁻⁵	47,58	37,94	46,17	36,5	45	36	5,73	5,39	2,6	1,38
61	27	9,3e ⁻⁵	6,7e ⁻⁵	51	40,2	48,8	39	48	38	6,25	5,79	1,67	2,63
65	27	1,05e ⁻⁴	5,01e ⁻⁴	44,14	31,29	44,14	31,20	43	30	2,55	4,3	2,65	4

Table 5. Counterflow results

COUNTERFLOW RESULTS													
Inlet temperature		Flow		Mathematical model		ANSYS FLUENT		Experimental tests		%Error mathematical model		%Error ANSYS	
T _{h, inlet} (°C)	T _{c, inlet} (°C)	V _h (m ³ /s)	V _c (m ³ /s)	T _{h, outlet} (°C)	T _{c, outlet} (°C)	T _{h, outlet} (°C)	T _{c, outlet} (°C)	T _{h, outlet} (°C)	T _{c, outlet} (°C)	Hot	Cold	Hot	Cold
45,5	28	1,905 e ⁻⁴	1,951 e ⁻⁴	39,71	33,60	38,55	34,10	38	33	4,50	1,81	1,44	3,33
50	28	1,947 e ⁻⁴	1,881 e ⁻⁴	42,1	34,39	40,84	34,1	40	33	5,25	4,21	2,10	3,33
55	28	1,243 e ⁻⁴	1,295 e ⁻⁴	44,15	37,33	42,98	36,43	42	36	5,11	3,69	2,33	1,19
60	28	1,031 e ⁻⁴	1,068 e ⁻⁴	45,11	39,45	44,11	39,5	44	39	2,52	1,15	0,25	1,28
65	28	1,040 e ⁻⁴	1,304e ⁻⁴	49,73	38,98	47,14	38,29	47	37	5,81	5,35	0,29	3,48

6. CONCLUSIONS

Based on the results obtained, it is possible to verify that the mathematical model proposed works properly, so it becomes a useful tool when evaluating and improving the design of the double tube heat exchanger.

Taking into account the assumptions that have been made in the mathematical model, such as the existence of fouling, the dependence of the value of the thermal properties of the temperature and the perfect isolation of the outer surface of the shell, there is a low margin of difference between the calculated values and the experimentally measured values.

The thermal resistance by conduction of the copper tube is very small in comparison with the resistances by convection, minimally affecting the effectiveness of the exchanger.

The results obtained showed an error lower than 6% for the mathematical model developed and 2% for the simulation in ANSYS with respect to the experimental results.

7. ACKNOWLEDGEMENTS

The authors gratefully acknowledge the financial support of CAPES, FAPEMIG and CNPq.

8. REFERENCES

- Arroyo. (2014). *Diseño bajo norma TEMA de intercambiadores de tubo y carcasa* . Madrid.
- Berrio, A. (2015). *Diseño de un sistema de monitoreo en labview aplicado a un banco de pruebas I.C de tubo* . Cúcuta .
- Bird, R. (2006). *Fenomenos de transporte* . Ciudad de México : Limusa Wiley .
- Boles, Y. C. (2013). *Termodinámica* . New York : McGraw-Hill .
- Ghajar, C. &. (2011). *Heat transfer and mass*. McGraw Hill .
- Gnielinski. (1976). *New Equations for heat and mass transfer in turbulent pipe and channel flow* .
- Incropera, F. (2015). *Fundamentos de transferencia de calor* . Prentice Hall.
- London, K. A. (3er Ed.). *Compact heat exchanger* . New York : McGraw Hill .
- Petukhov. (1972). *Heat transfer and friction in turbulent pipe flow with variable physical properties* . New York .

9. RESPONSIBILITY NOTICE

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