17th Brazilian Congress of Thermal Sciences and Engineering
November 25th-28th, 2018, Águas de Lindóia, SP, Brazil

ENCIT-2018-0239
SIMULATION OF A SHELL AND TUBE HEAT TRANSFER PROCESS WITH ANSYS

Tiago Henrique Santa Maria
Samuel Sander de Carvalho
Instituto Federal de Educação Ciência e Tecnologia do Sudeste de Minas
tiagostmec@gmail.com
samuel.carvalho@ifsudestemg.edu.br

Alexandre de Oliveira Mieli
UniRedentor
a_mieli@hotmail.com

Abstract. *The CFD (Computational Fluid Dynamics) is a branch of fluid mechanics that uses numerical analysis and data structures like FEA (Finite Element Analysis) to solve and analyze problems that involve the flow of fluids. This article uses the CFD to model and simulate a real shell and tube Heat Exchanger and compare the real results with the simulated ones to validate the CFD analysis.*

Keywords: *CFD, FEA, fluid flow, heat transfer, heat exchanger*

1. INTRODUCTION

The equations that describe fluid flow can be a bit intimidating, even though, we may limit ourselves to incompressible flows for which the viscosity is constant, we still end up with the following equations:

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}=0 \tag{1}$$

$$\rho(\frac{\partial u}{\partial t}+u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}+w\frac{\partial u}{\partial z}=\rho gx-\frac{\partial p}{\partial x}+\mu(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}+\frac{\partial^2 u}{\partial z^2}) \tag{2}$$

$$\rho(\frac{\partial v}{\partial t}+u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}+w\frac{\partial v}{\partial z}=\rho gx-\frac{\partial p}{\partial x}+\mu(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}+\frac{\partial^2 v}{\partial z^2}) \tag{3}$$

$$\rho(\frac{\partial w}{\partial t}+u\frac{\partial w}{\partial x}+v\frac{\partial w}{\partial y}+w\frac{\partial w}{\partial z}=\rho gx-\frac{\partial p}{\partial x}+\mu(\frac{\partial^2 w}{\partial x^2}+\frac{\partial^2 w}{\partial y^2}+\frac{\partial^2 w}{\partial z^2}) \tag{4}$$

Formerly, we have the Continuity Equation (1), followed by Navier-Stokes equations (2) (3) (4), expressed in Cartesian coordinates as shown above. In practice, there is no general analytic solution for these equations. Consequently, due to the combined effect of a number of reasons, the nonlinearity becomes a problem, and real prototypes may not have incompressible flows as well as other assumptions that were developed to use these equations fall apart. Then a question arises: Is there a way out for this issue?

Here is where we can use CFD (Computational Fluid Dynamics) softwares that present numerical methods to solve these real problems that do not show simple geometry and boundary issues (Fox, 2011).

CFD is a science commonly used to study the fluid flow, heat transfer, chemical reactions etc of a system by solving mathematical equations with the help of numerical analysis. It is equally helpful in designing a heat exchanger system from scratch as well as in troubleshooting/optimization by suggesting design modifications (Wang, 2007). CFD employs a very simple principle of resolving the entire system in small cells or grids and applying governing equations on these discrete elements to find numerical solutions regarding pressure distribution, temperature gradients, flow parameters in a shorter time and at a lower cost because of reduced required experimental work (Kanaris, 2006).

In this article we used a CFD software called ANSYS to model a laboratory heat exchanger with a shell and tubes that is shown in Figure 1 and compare the practical results with the simulated ones.

2. METHODOLOGY

2.1 Fluid Dynamics

Fluid dynamics is a science that studies the way to investigate flow patterns and physical results about the fluid in the case. This science has various applications in engineering, as we can see in the papers of Hoi et al. (2004) which analyses arterial sanguine flow, and Boutsioukis et al. (2009) which analyses irrigation flow for agricultural applications and the aim of his study was to investigate the effect of irrigant flow rate on the flow pattern within a prepared root canal during final irrigation with a syringe and needle using a CFD model.

Some other examples as in Fox (2011) include environmental and energy issues (e.g., containing oil slicks, large-scale wind turbines, energy generation from ocean waves, aerodynamics of large buildings, and fluid mechanics of the atmosphere, oceans and phenomena such as tornadoes, hurricanes, and tsunamis) biomechanics (e.g., artificial hearts and valves and other organs such as the liver, the understanding of the fluid mechanics of blood, synovial fluid in the joints, the respiratory system, the circulatory system, and the urinary system) sports (design of bicycles and bicycle helmets, skis, sprinting and swimming clothing, and the aerodynamics of golf, tennis, and soccer balls) “smart fluids” (e.g., in automobile suspension systems to optimize motion under all terrain conditions, military uniforms containing a fluid layer that is “thin” until combat, when it can be “stiffened” to give the soldier strength and protection, and fluid lenses with humanlike properties for use in cameras and cell phones) and micro fluids (e.g., for extremely precise administration of medications).

In this paper, fluid dynamics has its importance because it is used to analyse the fluid heat transfer potential and it is directly influenced by the flow pattern (laminar or turbulent). With the help of CFD we can model any problem and validate our results for a real implementation of the simulated problem. Despite the fact that CFD was originally developed for industrial and engineering purposes, in the last decade applications in the biomedical field have also attracted considerable attention (Xu et al. 2006), aided by the increasing power of computers (Politis et al. 2007). In this decade with the technology advances, computers that are even more powerful are used for these applications.

2.2 Heat Transfer

The importance of heat transfer and fluid flow is concerned with heat and mass transfer, fluid flow, chemical reaction, and other related processes that occur in engineering equipment, natural environment, and living organisms. These processes play a vital role that can be observed in a great variety of practical situations, since nearly all methods of power production involve fluid flow and heat transfer as essential processes (Patankar, 1980).

We have few methods called modes of heat transfer, they are called conduction, convection and radiation, the can be done in a fluid and in a solid (Incropera, 2011). And the heat transfer introduce us for heat exchanger problems.

2.3 Heat Exchanger

Heat exchangers are typically classified according to flow arrangement and type of construction. In this article the used assembly is “Shell and Tube”, with the big tube for cold flow and the small ones for hot flow. When hot and cold move in the same directions, it is called parallel-flow arrangement. On the other hand, when the fluids enter through opposite ports, flow in opposite directions and leave through opposite ends, it is called counter-flow arrangement. In this paper we used the shell and tube with counter-flow arrangement.

In the first experiment, the real heat exchanger was used to collect data about the cold inlet and outlet as well as the outlets to observe the real model results for later comparison. The results are shown in Table 1.

Table 1. Experimental results from the real heat exchanger.

	Temperature (K)	Flow Rate (L/min)
Hot Inlet	331,75	1,78
Hot Outlet	325,85	1,78
Cold Inlet	298,65	2,31
Cold Outlet	303,05	2,31

Then, with specific tools, measurements were taken to obtain the tube diameters and other piece of information such as their area. All this data would be later used to draw the shell and tube heat exchanger and for later simulations. We can see the measures on table 2.

Table 2. Measures from the real heat exchanger.

	Diameter (mm)	Area (m²)
Hot Internal	4	0,0000125
Hot External	6	0,0001979
Cold Internal (entrance/exit)	11	0,0000950
Cold External (entrance/exit)	13	X
Cold (big tube)	50	X

The shell and tube heat exchanger is a common configuration among other machines of this type. This one involves tubes and shell passes. Baffles are usually installed to increase the convection coefficient of the shell side fluid by inducing turbulence and a cross-flow velocity component. In addition, the baffles support the small tubes by reducing vibrations (Incropera, 2011).

The shell side flow is very complicated in shell-and-tube heat exchangers due to many different leakage paths and bypass streams between different flow zones. For different shell designs and sizes, the importance of each leakage and bypass streams may vary. However in small heat exchangers, these streams either do not exist or are negligible compared to the main flow stream (Ozden, 2010).

The baffles also act as turbulence promoters and provide support to tubes in horizontal units. Detailed knowledge of the flow, temperature, and turbulence fields within a heat exchanger help towards the design of reliable and efficient units. Experimental testing is usually expensive and time consuming. In addition, flow visualization and detailed turbulence measurements in heat exchangers are difficult to perform. A well-validated numerical model can serve as a cost-effective research tool for problems of shooting and design of shell-and-tube heat exchangers. To this end, a three-dimensional model has been developed for numerical simulation of shell-side flow and heat transfer in shell-and-tube heat exchangers (Prithiviraj, 1998).

For running successfully a full CFD simulation of a detailed heat exchanger model, large amounts of computing power and computer memory as well as long computation times are required. Without any simplification, an industrial shell-and tube heat exchanger with 500 tubes and 10 baffles would require at least 150 million computational elements, to resolve the geometry (Ozden, 2010). In this paper the heat exchanger used is utilized for educational purpose and that problem will not occur, but this avail may help us model an industrial version of this heat exchanger.

The real heat exchanger can be seen in Figures 2, 3 and 4. They show the real configuration of a shell and tube heat exchanger and the experiment can be performed to acquire the data that is shown in Table 1 above.

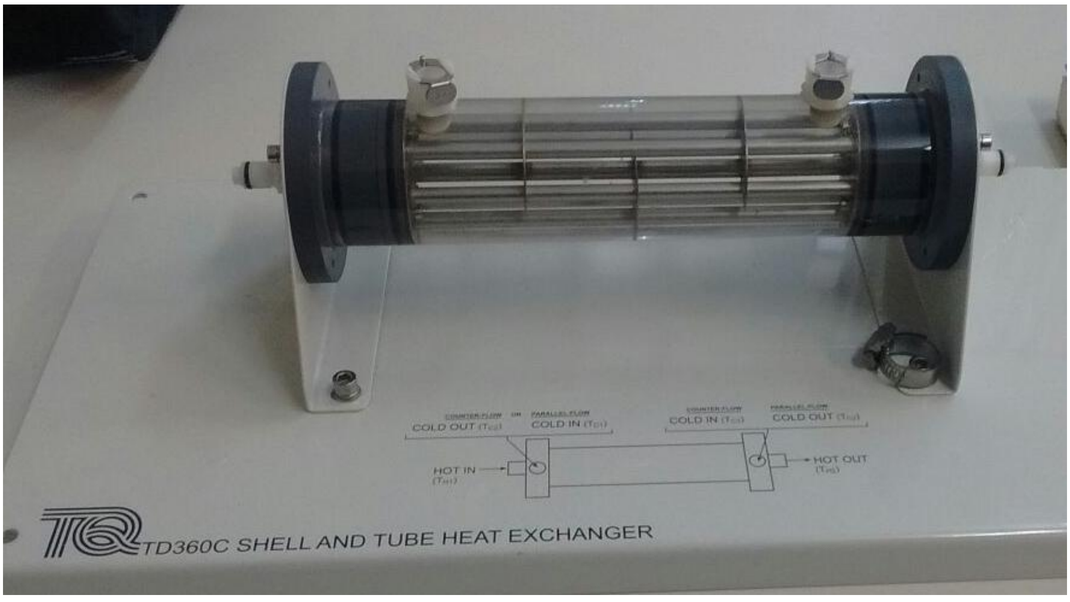

Figure 2. Shell and tube heat exchanger used in this work.

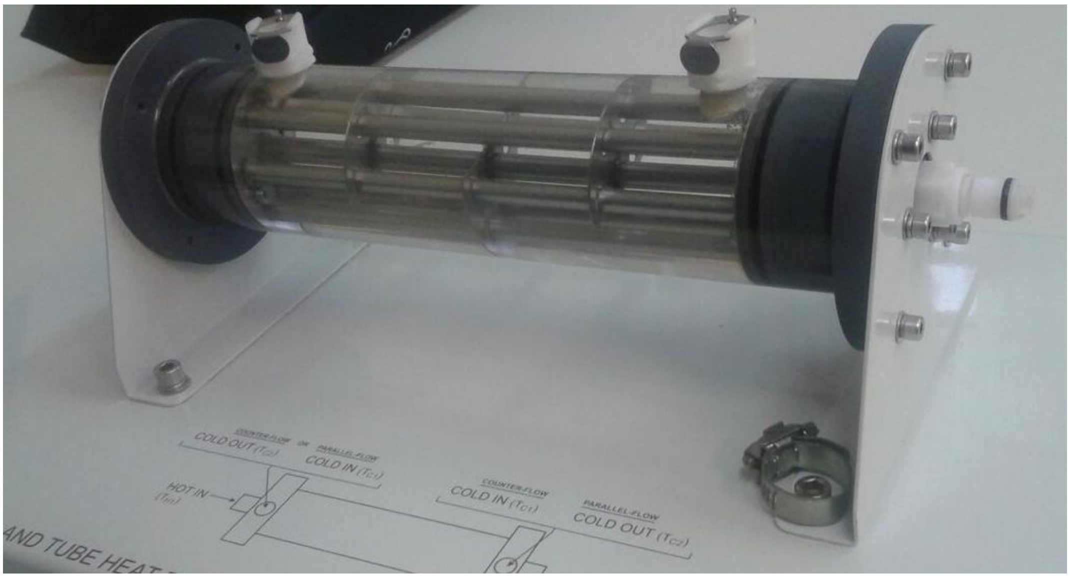

Figure 3. Shell and tube heat exchanger used in this work.

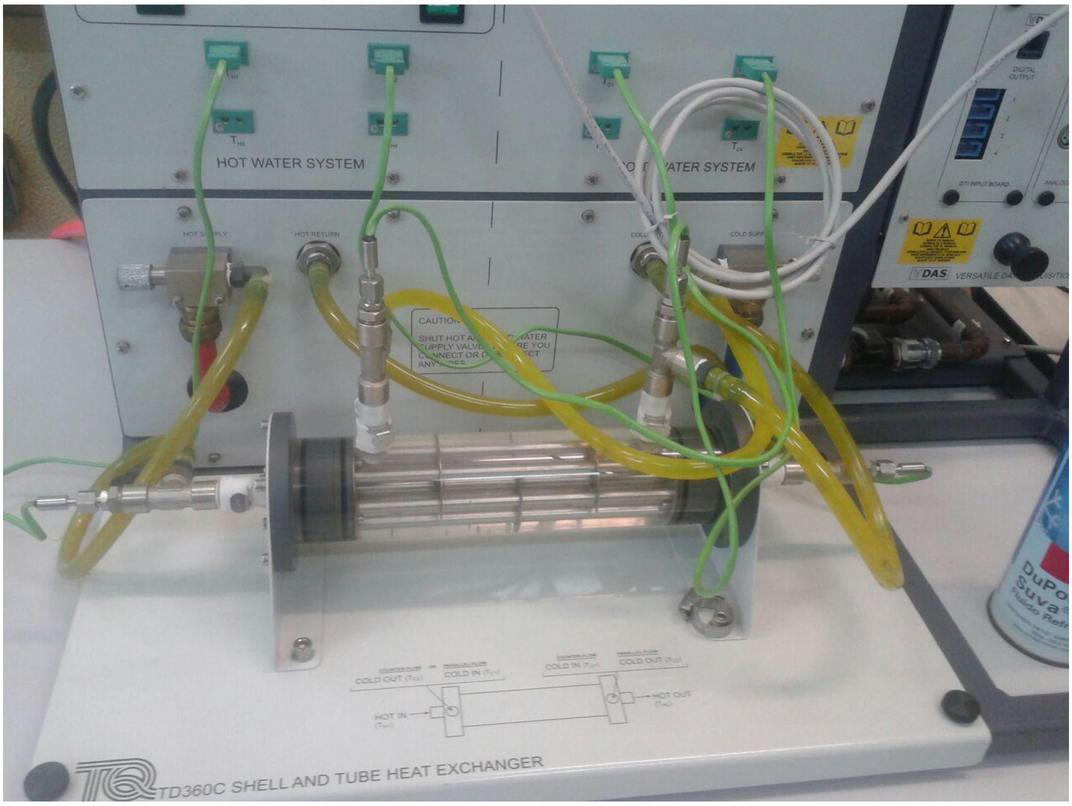

Figure 4. Shell and tube heat exchanger used in this work.

2.4 Ansys and CFD

CFD has also become one of the three basic methods or approaches that can be employed to solve problems in fluid dynamics and heat transfer. As demonstrated in Figure 5, each approach is strongly interlinked to each other and does not lie in isolation. Traditionally, both experimental and analytical methods have been used to study the various aspects of fluid dynamics and to assist engineers in the design of equipment and industrial processes involving fluid flow and heat transfer. With the advent of digital computers, the computational (numerical) aspect has emerged as another viable approach. Although the analytical method is still practiced by many and experiments will continue to be significantly performed, the trend is clearly towards greater reliance on the computational approach for industrial designs, particularly when the fluid flows are complex (Tu et al., 2018). In this article we used a Experimental fluid dynamics method to get results from the real heat exchanger and CFD to avail it.

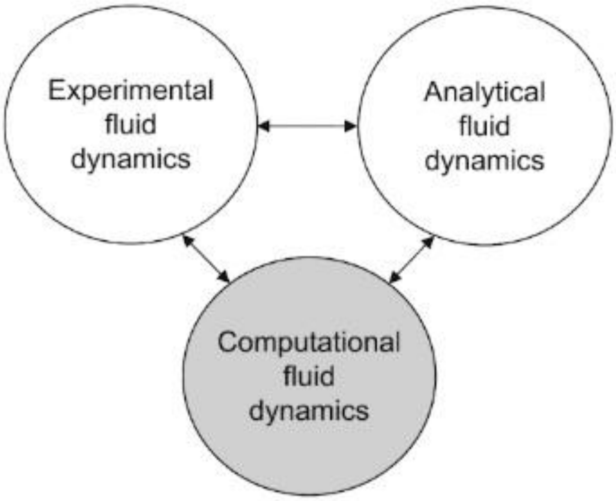

Figure 5. Analysis of a CFD problem (Tu, et al., 2018).

With the help of ANSYS Student Version 17/18 and the previous measures gathered from the real heat exchanger it was possible to start modelling the real heat exchanger using a CAD software. The result is in shown below in Figure 6.

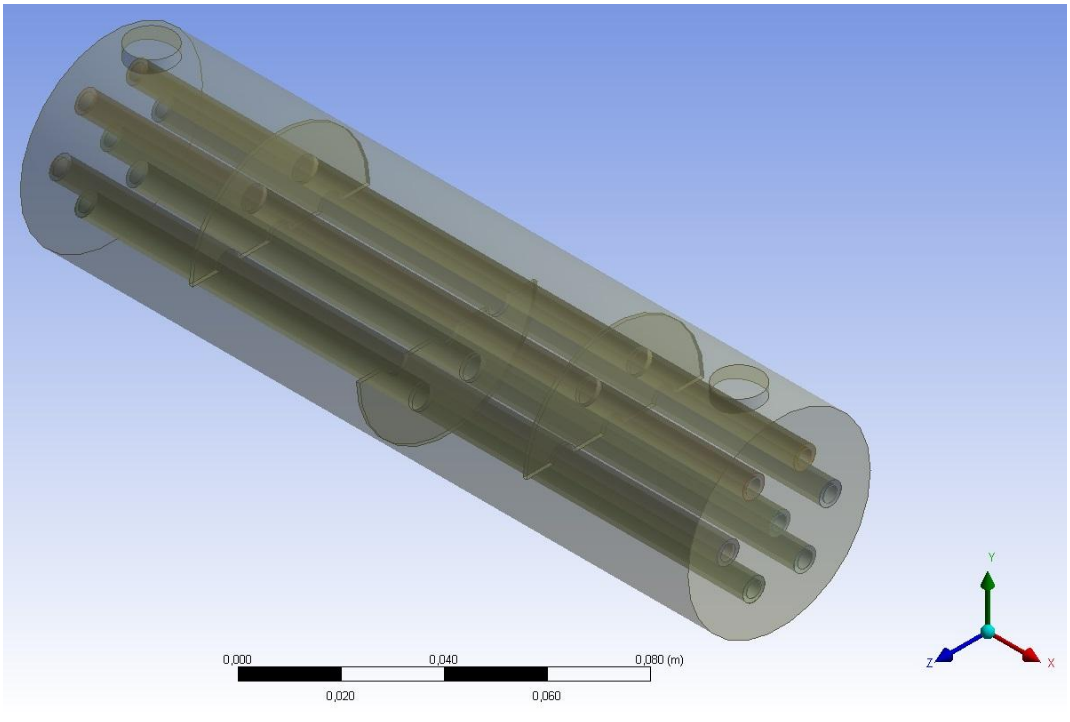

Figure 6. Model of the heat exchanger utilizing ANSYS DM.

Mesh or grid generations are an important part of a CFD analyses because they govern the partial differential equations, and the refinement of the results. Since ANSYS Student has its limitations about this part the number of nodes was limited and the results may not be optimal. The mesh used is polyhedral and coarsely adaptive. Figure 7 illustrates its appearance as a triangular mesh. All other settings are in the default settings.

What is a mesh in the context of CFD? By definition, a mesh itself consists of an arrangement of discrete number of points overlaying the whole domain geometry (Tu, et al., 2018).

Besides the polyhedral mesh, we have the structured mesh that has a uniform distribution of the points resulting in a rectangular geometrical form. The body fitted mesh has the same geometric form as the structured one but it adapts on the borders.

The unstructured meshes like the polyhedral one have advantages over the others, meaning a good option for fluids because of the curved boundaries, maximum flexibility and a faster convergence (Tu, et al., 2018).

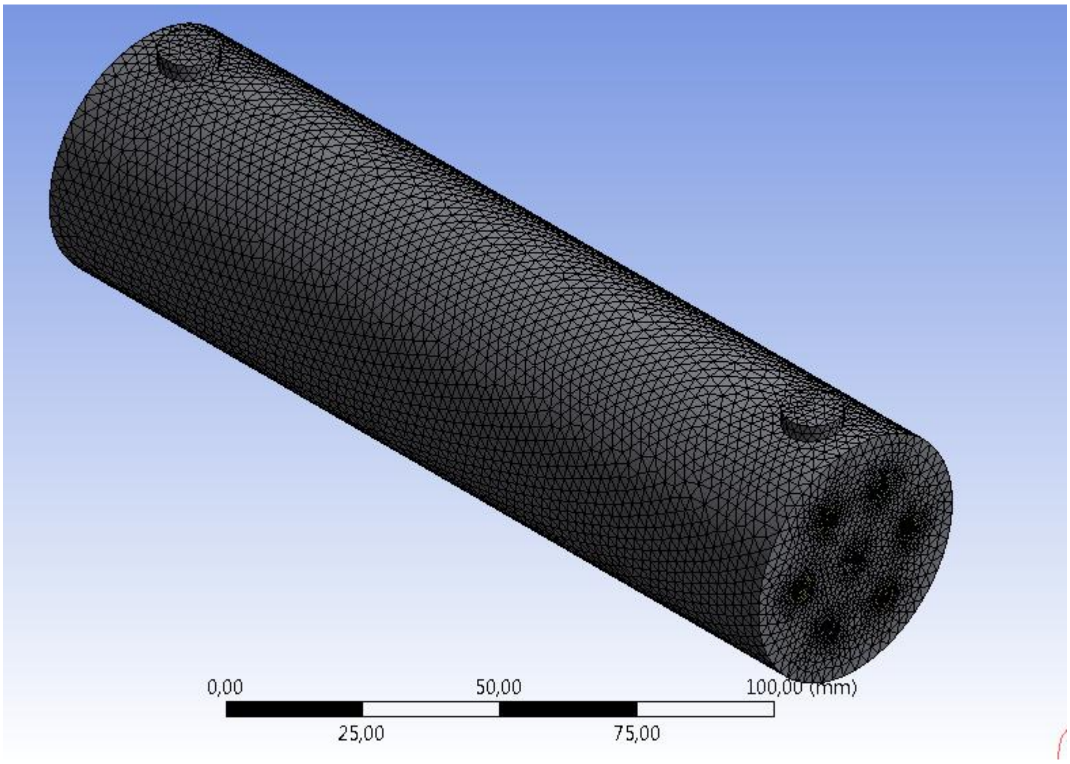

Figure 7. Example of a mesh of the heat exchanger .

The mathematical part of the job is to choose the right numerical model and put the correct values needed to simulate the heat exchanger like the hot and cold inlet temperatures, velocities and boundary conditions. We add the energy equation and choose a laminar flow (Reynolds number shows us that is possible). The solution method has velocity controlled inputs with Least Squares Cell Based gradient with second order momentums and energies. Therefore, once the real boundary conditions are created converting flow rate into velocity becomes necessary and it can be seen in Table 3. The materials utilized in the simulation are water and stainless steel 304 small tubes.

Table 3. Conversion of flow rate and calculated velocity.

	Flow Rate (L/min)	Flow Rate (m^3/s)	Velocity (m/s)
Hot Inlet	1,78	4,238e-6	0,3372
Hot Outlet	1,78	4,238e-6	0,3372
Cold Inlet	2,31	3,850e-5	0,4051
Cold Outlet	2,31	3,850e-5	0,4051

3. RESULTS

Table 4 provides the comparison between the results obtained in the real heat exchanger and the simulation ones.

The flow rate in both the simulated and the real version is the same, allowing us to compare the temperatures. The cold inlet is set as the ambient water temperature at 298,65 Kelvin, and the hot inlet is set at 331,75 Kelvin by a water heater before the control volume. By getting the practical results obtained with sensors and the simulated ones with the results above and comparing them, and we see a small amount of ΔT (difference from practical to simulated) as a satisfying result.

Another way to think about the results is close to the real parameters but it can be better if we utilize a new approach in CFD analysis or just try other heat exchanger methods like parallel flow to compare with these gathered results.

Table 4. REAL Final results VS SIMULATED Final results.

	Flow Rate (L/min)	T Practical (K)	T Simulated (K)	ΔT
Hot Inlet	1,78	331,75	331,75	0,00
Hot Outlet	1,78	325,85	327,29	1,44
Cold Inlet	2,31	298,65	298,65	0,00
Cold Outlet	2,31	303,05	301,14	1,91

Figures 8,9 and 10 show us a visual aspect of the results and how the heat transfers occur in the real heat exchanger. In figure 8, we can see the result of the simulation. The blue part clearly shows cold temperatures and we can see a light blue area in the exit of cold flow. The same goes for hot temperature. In the hot inlet a pure red color appears meanwhile in the outlet we can see the yellow/orange showing us that the fluid is cooling.

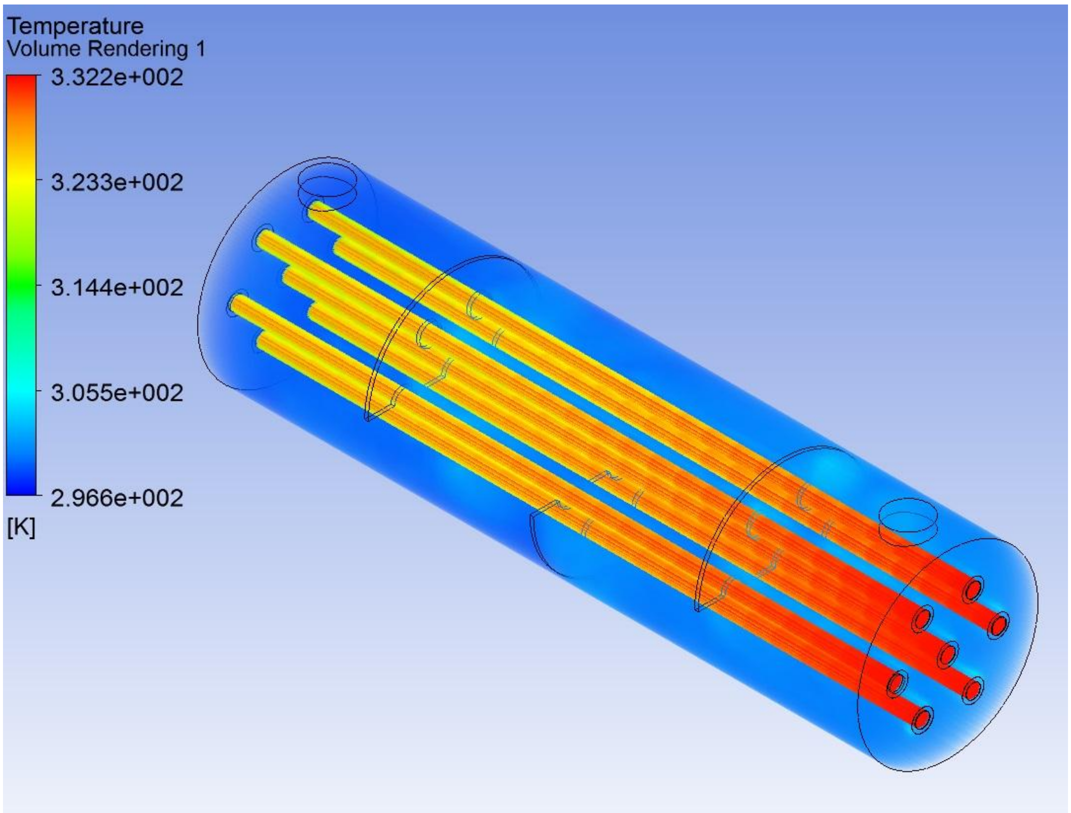

Figure 8. Simulation Volume Result .

Figure 8 is a generic way to look at the results. They can be seen but there is not too much to say about it, since the entrance (left corner) is bluer or colder and in the big tube a lighter blue in the exit (right corner) indicates that the fluid is hotter than it is in the entrance like it should be. The same goes with the hot tubes. The red in the entrance (right corner) is more intense and gradually turns into yellow near the exit. This shows the heat exchanger is working in the counter-flow mode.

Figure 9 is a contour of the cold results. The result obtained is satisfactory in general but the most important result in this article is the exit and not the complete system. Finally, figure 10 gives us the same vision of figure 9 but for the small inlets of hot fluids.

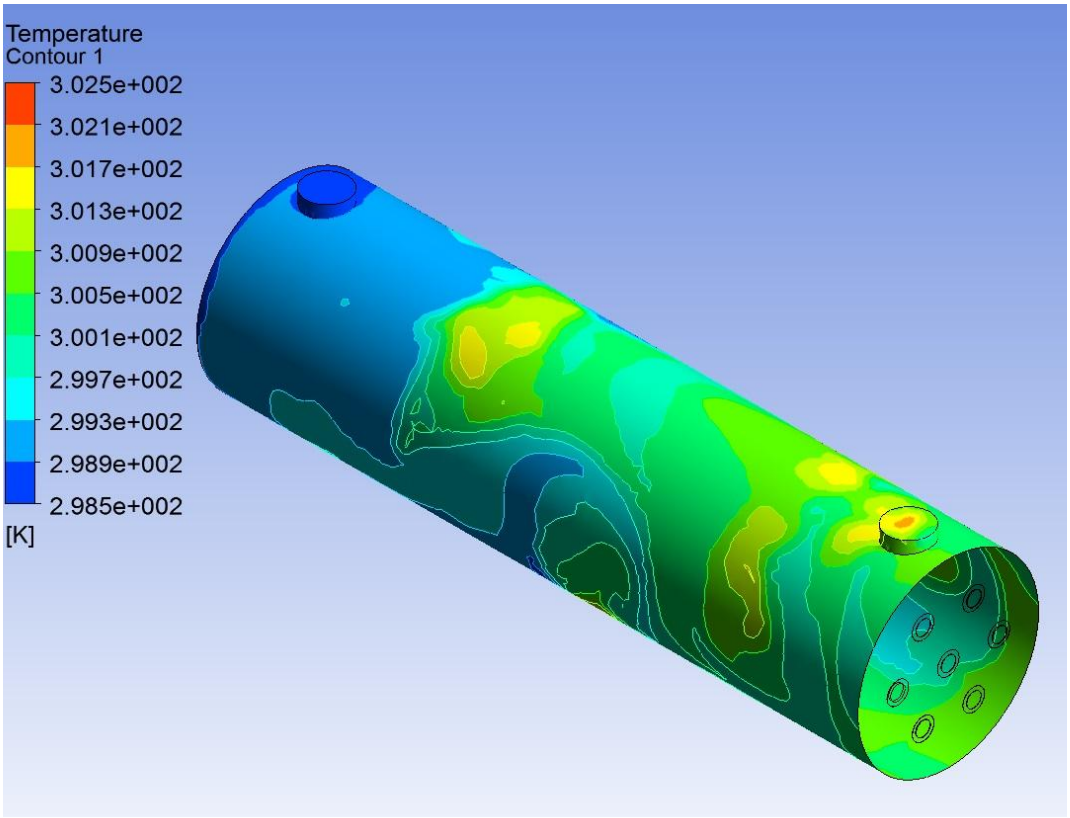

Figure 9. Simulation outside cold contour result .

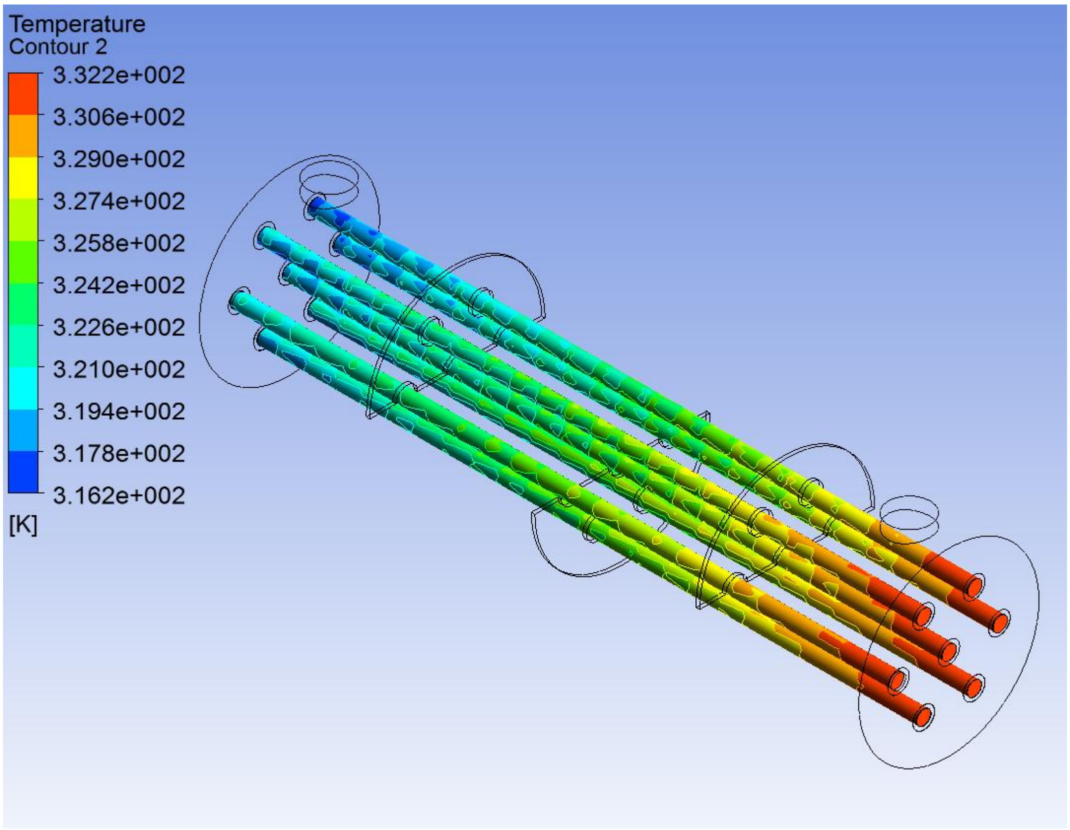

Figure 10. Simulation outside contour hot result.

4. CONCLUSIONS

The final results are satisfying since the ΔT in Kelvin is pretty small and the utilized software is a student version which has limitations on its nodes and elements of the mesh. With a refined mesh on a PRO version of ANSYS the results may be closer to the real parameters**.** Another situation that differs from reality and, consequently, directly influences the accuracy of the result is that the real heat exchanger may have some load loss that was not considered in the simulated process due to the limitations of the software.

The load loss problem can be real since the accuracy of the computational solutions can be affected by errors and uncertainties in the numerical calculations. These errors and uncertainties can be generated either in the conceptual modeling or during the design phase. They need to be measured and bounded. The credibility of the solution is strongly dependent on whether the errors and uncertainties are indentified and qualified (Tu, et al., 2018).

The creators of this article give their thanks to Instituto Federal de Educação Ciência e Tecnologia do Sudeste de Minas - Campus Juiz de Fora for all the support in the content.

5. REFERENCES

Boutsioukis, C., T. Lambrianidis, and E. Kastrinakis. "Irrigant flow within a prepared root canal using various flow rates: a computational fluid dynamics study." *International Endodontic Journal* 42.2 (2009): 144-155.

Fox, R. W. and Mcdonald A. T., 2011. “Introduction to fluid mecanics”.

Hoi, Yiemeng, et al. "Effects of arterial geometry on aneurysm growth: three-dimensional computational fluid dynamics study." *Journal of neurosurgery* 101.4 (2004): 676-681.

Incropera, F., 2011. “Fundamentals of Heat Transfer and Mass”.

Kanaris, Athanasios G., Aikaterini A. Mouza, and Spiros V. Paras. "Flow and heat transfer prediction in a corrugated plate heat exchanger using a CFD code." *Chemical engineering & technology* 29.8 (2006): 923-930.

Ozden, Ender, and Ilker Tari. "Shell side CFD analysis of a small shell-and-tube heat exchanger." *Energy Conversion and Management* 51.5 (2010): 1004-1014.

Patankar, Suhas. *Numerical heat transfer and fluid flow*. CRC press, 1980.

Prithiviraj, M., and M. J. Andrews. "Three dimensional numerical simulation of shell-and-tube heat exchangers. Part I: foundation and fluid mechanics." *Numerical Heat Transfer, Part A Applications* 33.8 (1998): 799-816.

Politis AK, Stavropoulos GP, Christolis MN, Panagopoulos FG, Vlachos NS, Markatos NC (2007) Numerical modeling of simulated blood flow in idealized composite arterial coronary grafts: steady state simulations. Journal of Biomechanics 40, 1125–36.

Tu Jiyuan, Guan-Heng Yeoh, and Chaoqun Liu. *Computational fluid dynamics: a practical approach*. Butterworth-Heinemann, 2018.

Wang, Yongqing, Qiwu Dong, and Minshan Liu. "Characteristics of Fluid flow and heat transfer in Shellside of Heat Exchangers with Longitudinal Flow of Shellside Fluid with Different Supporting structures." *Challenges of Power Engineering and Environment*. Springer, Berlin, Heidelberg, 2007. 474-479.

Xu C, Sin SH, McDonough JM et al. (2006) Computational fluid dynamics modeling of the upper airway of children with obstructive sleep apnea syndrome in steady flow. Journal of Biomechanics 39, 2043–54.

6. RESPONSIBILITY NOTICE

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