Simulation of a tube and shell heat exchanger with COMSOL-Multiphysics®

Simulación de un intercambiador de calor de tubo y coraza con COMSOL-Multiphysics®

MARTÍNEZ-VÁZQUEZ, J. Merced†´, CORTÉS-CAMPOS, M. de Lourdes*´´, HORTELANO-CAPETILLO, J. Gregorio´ and VARGAS-RAMÍREZ, Marissa´´´

´Universidad Politécnica de Juventino Rosas, Metallurgical Engineering. Hidalgo 102, Comunidad de Valencia, Santa Cruz de Juventino Rosas, Gto. 38253, Mexico.
´´Universidad Politécnica de Guanajuato, Agro-industrial Engineering. Avenida Universidad Norte s/n, Cortazar, Gto. 38294, Mexico.
´´´Universidad Autónoma del Estado de Hidalgo-ICBI, Ciencias de la Tierra y Materiales, Carretera Pachuca-Tulancingo Km. 4.5, Pachuca de Soto, Hgo. 42090, Mexico

ID 1st Author: *J. Merced, Martínez-Vázquez* / **ORC ID:** 0000-0002-6230-3846, **CVU CONACYT ID:** 93450

ID 1st Co-author: *M. de Lourdes, Cortés-Campos* / **ORC ID:** 0000-0003-1267-25603

ID 2nd Co-author: *J. Gregorio, Hortelano-Capetillo* / **ORC ID:** 0000-0002-3702-4853, **CVU CONACYT ID:** 347496

ID 3rd Co-author: *Marissa, Vargas-Ramírez* / **ORC ID:** 0000-0002-5968-6196

DOI: 10.35429/JEA.2022.27.9.17.22 Received: January 30, 2022; Accepted: June 30, 2022

Abstract

Shell and tube heat exchangers are ideal for applications with extreme conditions of temperature, pressure and corrosive fluids. Its conformation consists of tubes, tube sheets, baffles, channel covers, connections and shell; these can take various forms, which will depend on the desired task, pressure drop, thermal stress, the need to prevent leaks, pressures, temperatures, fluid corrosion, and maintenance, among other factors. Each of the exchangers used in industries have advantages and disadvantages, for example; evaporators, vaporizers and boilers have almost the same operation since they work with liquid fluids that are superheated in order to convert the fluid into steam, but it should be mentioned that in the case of the first two, they work with liquid fluids of different nature. This paper shows the results obtained from the simulation with the COMSOL-Multiphysics® program in order to establish the appropriate variables for the operation of a heat exchanger, in addition to comparing the theoretical results and the results obtained with the simulator.

Heat exchanger, Shell and tuve, simulation

Resumen

Los intercambiadores de calor de tubo y coraza son ideales para aplicaciones de condiciones extremas de temperatura, presión y fluidos corrosivos. Su conformación consiste de tubos, placas tubulares, deflectores, cubiertas de canal, conexiones y carcasa; estos pueden ser de diversas formas, que dependerán del cometido deseado, caída de presión, esfuerzos térmicos, necesidad de prevención de fugas, presiones, temperaturas, corrosión de los fluidos, y mantenimiento, entre otros factores. Cada uno de los intercambiadores utilizados en las industrias presentan ventajas y desventajas, por ejemplo; los evaporadores, vaporizadores y calderas tienen casi el mismo funcionamiento ya que estos trabajan con fluidos líquidos que son sobrecalentados con el fin de convertir al fluido en vapor, pero hay que mencionar que en el caso de los dos primeros se trabaja con fluidos líquidos de distinta naturaleza En el presente trabajo se muestran los resultados obtenidos de la simulación con el programa COMSOL-Multiphysics® con la finalidad de establecer las variables adecuadas para la operación de un intercambiador de calor, además de comparar los resultados teóricos y los resultados obtenidos con el simulador.

Intercambiador de calor, Tubo y coraza, simulación

Citation: MARTÍNEZ-VÁZQUEZ, J. Merced, CORTÉS-CAMPOS, M. de Lourdes, HORTELANO-CAPETILLO, J. Gregorio and VARGAS-RAMÍREZ, Marissa. Simulation of a tube and shell heat exchanger with COMSOL-Multiphysics®. Journal of Engineering Applications. 2022. 9-27:17-22.

* Correspondence of the Author (Email: mcortes@upgto.edu.mx)
† Researcher contributing as first author.

Introduction

It is well known that the methodological procedure to solve a problem in engineering consists of representing it in a correct and precise way, in order to obtain a substitution of the real system by a more adequate one for the formal treatment [1]. In general, the logical-mathematical tools provide a useful framework to represent by means of a system of symbols and rules, the behavior of real systems.

The scientific method helps us to consolidate laws and theories in different areas of knowledge, which are understandable by means of differential equations, for example. In other words, we are able to recreate a new system, of which we know its rules and symbols, such as of which we know its rules and symbols, as the result of a process of abstraction from the environment.

Process simulation using computer software has become popular among researchers. However, publications related to this field using open source as its simulation platform are yet to be seen [2]. This paper shows the results obtained with software simulation of a shell and tube heat exchanger using COMSOL Multiphysics® [3].

Shell and tube heat exchangers are one of the most widely used types of heat exchangers in industries with approximately 65% of the market [4]. They are found in oil refineries, nuclear power plants and other large-scale chemical processes. In addition, they can be found in many engines where they are used to cool hydraulic fluids and oil. In this type of equipment, two separate fluids at different temperatures flow through the heat exchanger: one through the tubes (tube side) and the other through the shell around the tubes (shell side) [5], as seen in Figure 1, so in their design it is necessary to consider the influence of various parameters and operating conditions to achieve their optimum performance.

The main objective of this work is to show the basic principles to configure a heat exchanger model that serves as a starting point for more sophisticated projects both for more complex applications, study of parameters or add additional effects such as corrosion, thermal stress and vibration among others [6].

The resolution of heat exchanger problems, although it can be done manually with the use of appropriate formulas, can also be facilitated by some specific programs, an example of this is the COMSOL program, which allows the simulation of thermal equipment generally used in industries [7], so, for this reason the program is useful for solving problems by correctly establishing the variables in order to obtain accurate results that show the temperature profile of the hot and cold fluids, or the total amount of heat conduction or convection [8].

Description of the model

The procedure used to design a shell and tube heat exchanger was performed by defining the straight, cross-flow, single-pass shell and tube heat exchanger [9].

The heat exchanger is made of mainly structural steel. In this work, two fluids at different temperature were considered, the first one is cold water passing through the tubes, while the second fluid, hot water, circulates inside the heat exchanger shell, but outside the tubes. Both fluids have different initial temperatures; however, after circulating inside the shell and tubes, the fluids approach an equilibrium temperature. The use of baffles introduces turbulence and cross-flow in the air thus increasing the heat exchange area. Another advantage is that the baffles reduce vibration due to fluid motion.

This model uses the physical interface of non-isothermal flow together with the k-ε turbulence model. It takes advantage of symmetries to model only half of the heat exchanger, which reduces model size and computational costs.

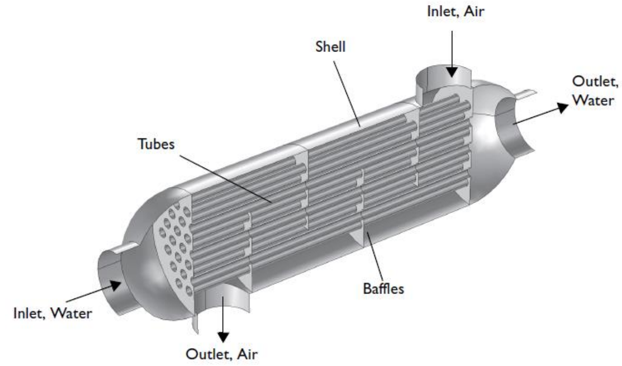


Figure 1 Schematic of a shell and tube heat exchanger

Article
Journal of Engineering Applications
June 2022, Vol.9 No.27 17-22

The condition taken for the simulation was that all the walls of the heat exchanger are modeled as shells. This requires special boundary conditions for the heat flow and heat transport equations [6].

To account for the in-plane heat flow in the shell, the boundary condition of the highly conductive layer is applied:

$$-\boldsymbol{n}\cdot(-k\nabla T) = -\nabla_t \cdot (-d_s k_s \nabla_t T) \qquad (1)$$

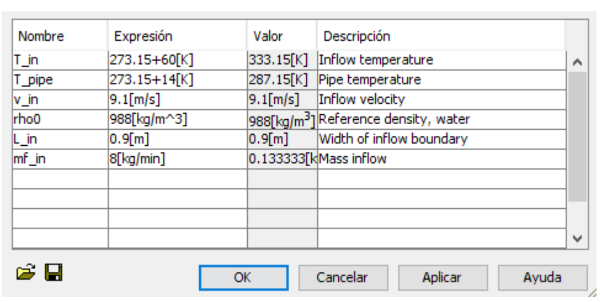
Here, ∇_t is the tangential derivative, ds is the layer thickness and ks is the thermal conductivity of the cover.

Methodology

The problem statement is as follows:

In a chemical plant a certain chemical product is heated by means of hot water supplied by a boiler in which natural gas is burned. The hot water (C_p = 4180 J/kg·°C) is then discharged at 60°C, at a rate of 8 kg/min. The average temperature of the cold water entering the boiler throughout the year is 14°C. In order to save energy, it is proposed to install a heat exchanger to preheat the cold water entering via the drained hot water. If it is assumed that the exchanger will recover 72% of the heat available in the hot water, determine the nominal heat transfer capacity of the exchanger that needs to be purchased.

The solution process was based on a COMSOL gallery model, to which the necessary constants were added, changing only the fluid temperatures; as was the mass flow rate, which was 8 kg/min (Figure 2).



Nombre	Expresión	Valor	Descripción
T_in	273.15+60[K]	333.15[K]	Inflow temperature
T_pipe	273.15+14[K]	287.15[K]	Pipe temperature
v_in	9.1[m/s]	9.1[m/s]	Inflow velocity
rho0	988[kg/m^3]	988[kg/m^3]	Reference density, water
L_in	0.9[m]	0.9[m]	Width of inflow boundary
mf_in	8[kg/min]	0.133333[k	Mass inflow

Figure 2 COMSOL image where the initial conditions are established

Then, the subdomain values were added to establish the type of material; in this work, copper was used due to its good qualities as a heat conductor (Figure 2).

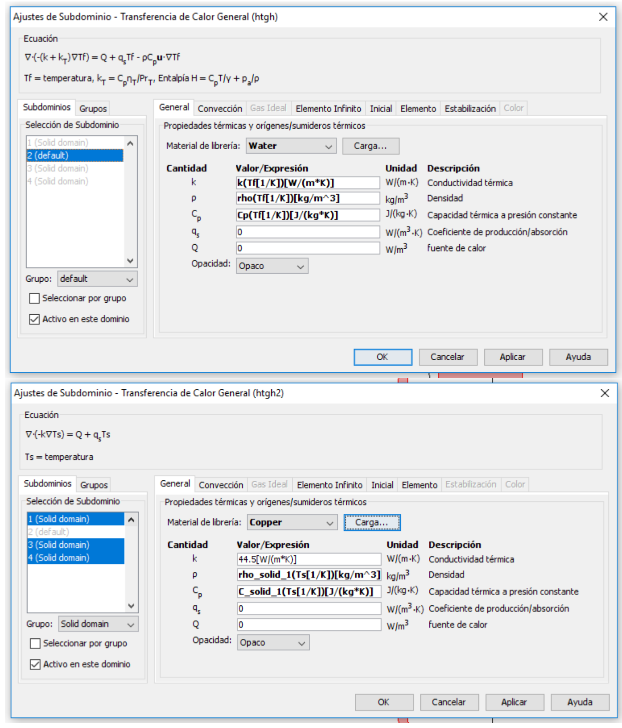


Figure 3 Subdomain conditions and type of material

Subsequently, contour values were established, this is where the walls that are going to be conductive and those that remain as insulators are determined (Figure 4).

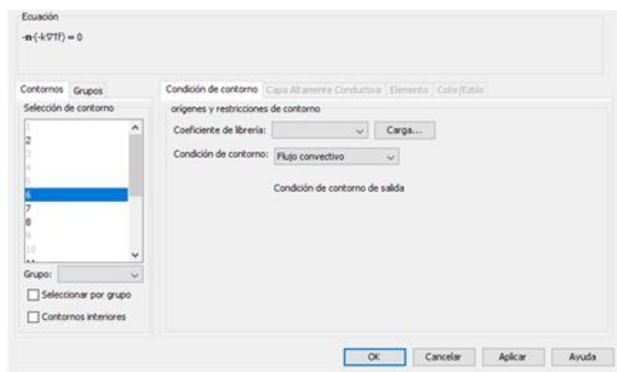


Figure 4 Boundary values conditions.

As a next step, the type of flow was established (it can be heat transfer by conduction, convection and radiation), in this work a convective flow was selected in the fluid outlet area of the shell (Figure 5).

A temperature of 60°C was set at the shell inlet (Figure 6). The walls marked in red were set as the cold zones of the exchanger (Figure 7).

Finally, the exchanger wall was considered as insulation (Figure 8).

ISSN: 2410-3454

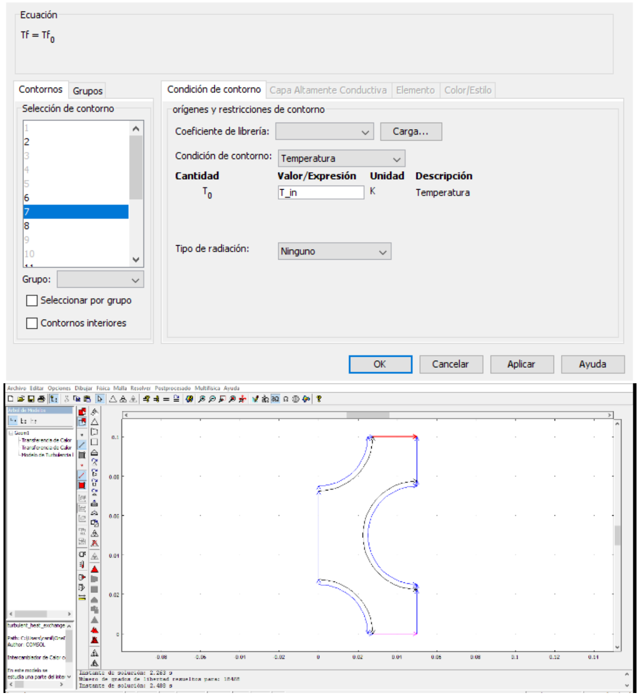


Figure 5 Selection of the type of convective flow, the blue lines indicate the zone of the exchanger where this condition will be

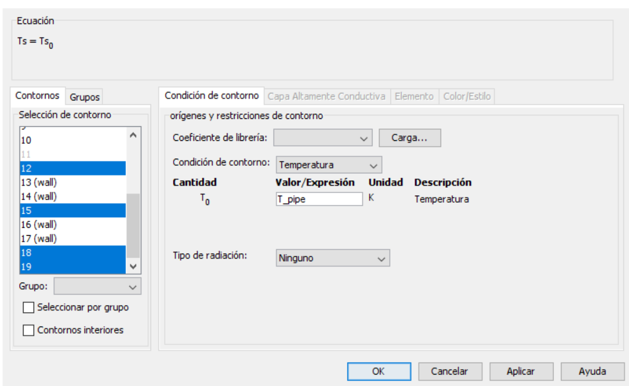


Figure 6 60°C temperature at the casing inlet

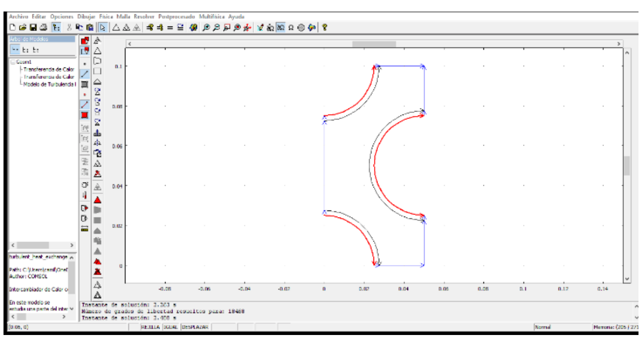


Figure 7 The red lines indicate the cold zones of the exchanger

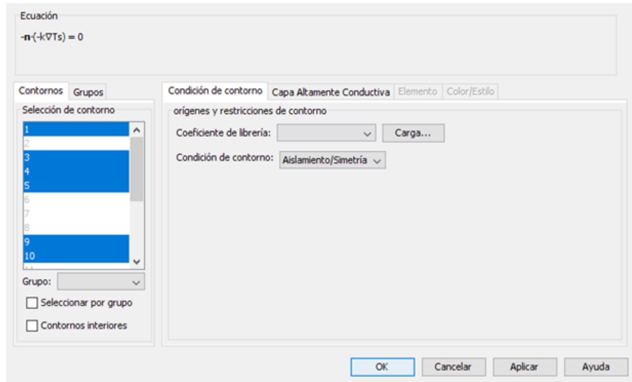


Figure 8 The walls of the exchanger were considered as insulators

Results

The theoretical calculations are shown below.

$$\dot{Q}_{max} = \dot{m}_h C_{ph}\left(T_{h,in} - T_{c,in}\right)$$

$$\dot{Q}_{max} = \left(\frac{8}{60}\frac{kg}{s}\right)\left(4.18\frac{KJ}{Kg}\times °C\right)(60°C - 14°) = 25.6\ kW$$

$$\dot{Q} = \varepsilon * \dot{Q}_{max} = (0.72)\left(25.6\frac{KJ}{s}\right) = 18.43\ kW$$

The simulation results are as follows:

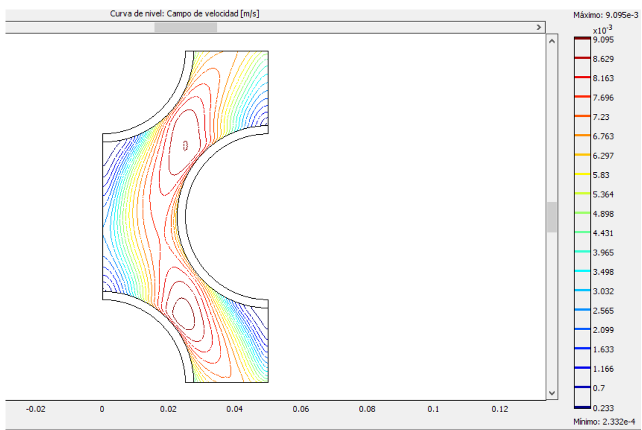


Figure 9 Fluid velocity profile in the housing

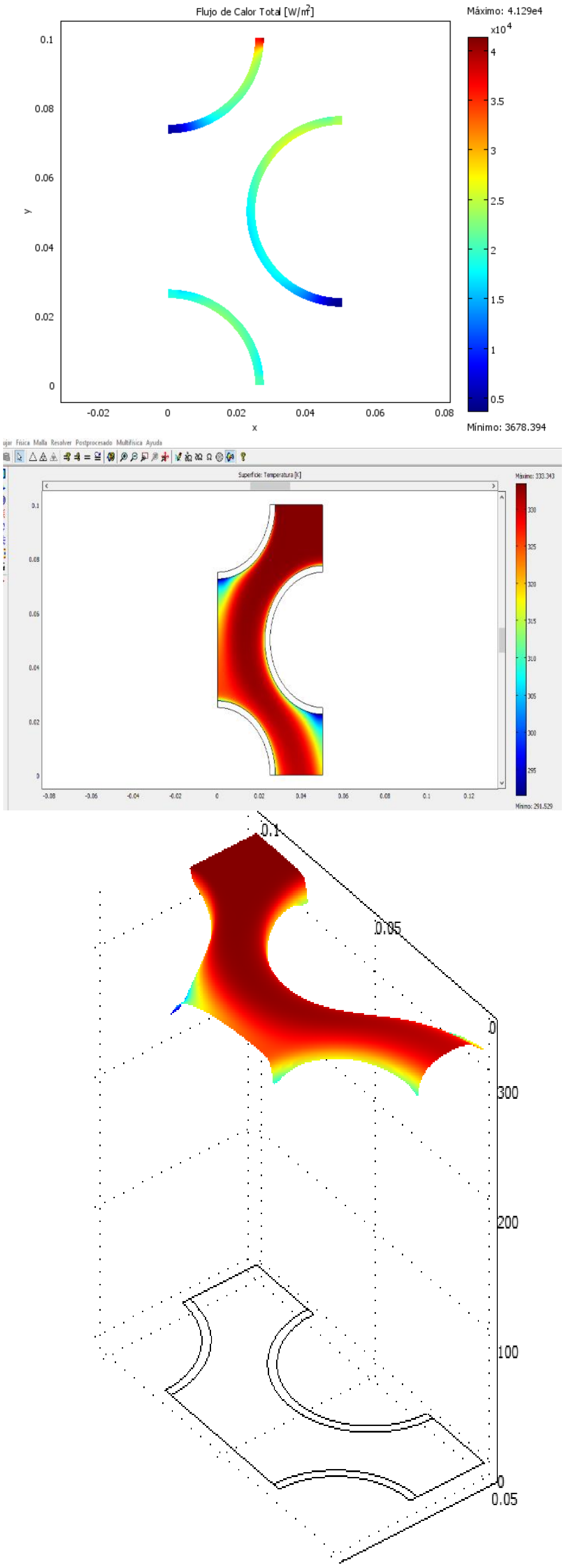


Figure 10 Schematic representations of the conductive heat fluxes in the tubes, the column shows the variation of colors according to the range of heat flow

The water current flowing through the shell of the exchanger corresponds to the hot fluid, whose inlet temperature is 60 °C and through the tubes of the exchanger flows the cold water, which is the one that requires to be preheated, the graph obtained from the simulation in COMSOL shows how the outlet temperature of the hot fluid from the shell decreases, becoming fainter color with respect to the color that acquires the higher temperature. On the other hand, the temperature that could be observed outside the tubes presented different shades, which represent the different temperatures that occur due to the effect of the velocity profile where the cold water flows, in addition, outside these tubes it was observed that the heat transfer was carried out by conduction due to the direct contact of the hot stream with the tubes.

The heat fluxes ranged from 17000 to 18800 W/m^2, considering, how in the theoretical calculations that the heat exchange area was 2.54 m2, the total amount of heat transfer was in the range of 43,180 kW to 47,752 kW. That is, almost 2.5 times that obtained theoretically. The difference is that the theoretical calculations do not consider the velocity profiles of the fluids, the shape and arrangement of the tubes, the temperature profile originated by the movement of the fluids, in addition to the type of heat flow, which in the simulation was conductive and convective, therefore, the value of the simulation would be closer to that obtained in a real operation of the heat exchanger.

Conclusions

In accordance with the objective of the project, the variables that were considered adequate in the description of the problem were established and assigned, and when obtaining the resolution of the problem in the program, it was possible to observe graphically the representation of the profiles of temperature, velocity and heat flow at the inlet and outlet of the fluid in the tubes and casing. This allowed to obtain a different result, however, more accurate to the real operation of this type of equipment.

Acknowledgements

The authors would like to thank UPJR and UPGTO for the facilities provided for this work.

References

[1] N. J. Scenna et al. **(1999)**. *Modelado, simulación y optimización de procesos químicos,* Argentina, Ed. Nicolás José Scenna. ISBN: 950-42-0022-2

[2] M. N. Omar et al. **(2018).** *Simulation comparison on PRICO LNG process using open and proprietary sources,* ICSCA, ICSCA 2018: Proceedings of the 2018 7th International Conference on Software and Computer Applications, February 2018, pp. 36–39. DOI: 10.1145/3185089.3185106

[3] H. S. Lee **(2010)**. *Thermal Design*, Ed. John Wiley & Sons. Canadá. ISBN 978-0-470-49662-6

[4] M. Vajdi et al. **(2020)**. A *review on the Comsol Multiphysics studies of heat transfer in advanced ceramics*, Journal of composites and compound, 2, 35-43. DOI: 10.29252/jcc.2.1.5

[5] J. A. Rangel Arista et al. **(2015).** *Simulación numérica de un intercambiador de calor Concéntrico para aplicaciones geotérmicas*, 12º Congreso Iberoamericano de Ingeniería Mecánica, volumen 12, Guayaquil, Ecuador. [https://www.researchgate.net/publication/290435356_Simulacion_numerica_de_un_intercambiador_de_calor_concentrico_para_aplicaciones_geotermicas] Revisado el 28 de septiembre de 2022

[6] T. K. Nguyen et al. **(2021)**. *Simulation of shell and tube heat exchanger using COMSOL software*, IOP Conf. Ser.: Earth Environ, Sci. 9471007. DOI: 10.1088/1755-1315/947/1/012008

[7] Andriawanet al. **(2020)**. *Simulation and Study of Shell and Tube Type Heat Exchangers*, IOP Conf. Series: Materials Science and Engineering 1007. DOI: 10.1088/1757-899X/1007/1/012179

[8] A. A. Paredes-Álvarez et al. **(2015)**. *Modelamiento y simulación de un intercambiador de calor*, Repositorio Escuela Superior Politécnica del Litoral, enlace del recurso. [http://www.dspace.espol.edu.ec/xmlui/handle/123456789/30673] Revisado el 19 de julio de 2022.

[9] J. G. Álvarez et al. **(2020)**. *Control de temperatura en intercambiadores de calor tipo coraza-tubo: una revisión realizada a la industria,* Entre Ciencia e Ingeniería, 14, pp. 41-49. DOI: 10.31908/19098367.1763.