

Design of Shell and Tube Heat Exchanger Articulating Analytical and Numerical Simulation Solutions

Juan P. Cardona^{1*} , José E. Ustariz² 

1. Universidad Cooperativa de Colombia, School of Engineering, Av. Caracas No. 37-15, Bogotá D.C. Colombia.
2. Universidad Cooperativa de Colombia, School of Engineering, Av. Caracas No. 37-15, Bogotá D.C. Colombia;
jose.ustariz@campusucc.edu.co

* Correspondence: juan.cardonag@campusucc.edu.co

Abstract: The thermal design of a shell and tube heat exchanger in this paper is performed by articulating analytical and computational numerical simulation solutions. The first uses the logarithmic mean temperature difference (LMTD) method to calculate the heat transfer and mass flow rates for a countercurrent arrangement of the heat exchanger and serves as input for the numerical simulations which are based on CFD computational fluid dynamics and are developed using SolidWorks software and its Flow simulation plug-in. Simulation results validate the analytical solutions and present the evolution of the temperatures and velocities of the hot and cold fluids inside the heat exchanger in the form of shear plots and flow paths, as well as the external shell temperature and the internal temperature of the tubes, baffles and separators are shown in surface plots.

Keywords: Heat exchanger; Computational fluid dynamics; Simulation; Thermal design

1. Introduction

As a result of the second law of thermodynamics when there is a temperature difference between two close bodies there is energy flow in the form of heat from the body of higher temperature to the lower one, the heat transfer between them can never stop, but can only slow down while they reach thermal equilibrium, and can only occur through three mechanisms which are conduction, convection and radiation or any combination of these [1]. One of the main applications of transfer processes are heat exchangers which are mechanical devices that are used to transfer the thermal energy of two fluids, and they are the most commonly used equipment in the industries of refrigeration, air conditioning, space heating, petrochemical plants, chemical plants, power plants, and natural gas processes[2]. It can be recognized that the use of any process involving heating, boiling, evaporation, cooling, or condensation will require a heat exchanger for these purposes[3]. The sole purpose of heat exchangers is to improve the heat transfer between the two fluids, thus reducing energy requirements and helping to make the process more efficient [4].

One of the most widely used heat exchangers in the industry is the shell and tube type which consists of a tube bundle enclosed within a cylindrical shell, its basic operating principle is that two fluids flow at different temperatures separated by a wall. The former flows through the tubes and the latter within the space between the tubes and the shell; due to the temperature difference between the fluids, heat transfer occurs from the higher temperature fluid to the lower temperature fluid by the mechanism of conduction and convection. The flow on the shell side is quite complicated. The fact that the inlet and outlet are perpendicular to the general flow direction creates complexities in shell flow[3]. For [5], this type of equipment is one of the most widely used in the industry because they have larger heat transfer surface area to volume ratios than most common types of exchangers and are easily manufactured for a variety of sizes and flow configurations; moreover, they can operate at high pressures and their construction facilitates disassembly for maintenance and cleaning periods[6]. Within shell and tube heat exchangers, the most common type is the one with baffles, which are plates that support the tubes and displace the flow between the inside and outside of the shell side. Such equipment must be compact, lightweight, and with high efficiency. Authors such as[2] point out the importance of selecting the right type of exchanger for a particular process because the incorrect selection of this causes undesired operation and equipment failures. The design procedure (analytical solution) of heat exchangers is quite complicated as it needs accurate analysis of heat transfer rate, efficiency, and pressure drop in addition to issues such as long-term performance and the economical aspect of the equipment[7].

According to [8, 9, 10], the Logarithmic Mean Temperature Difference (LMTD) method is presented as an analytical solution for the thermal design of the heat exchanger, which relates the transfer area, heat transfer, and inlet and outlet temperatures of the working fluids using four simplifications: 1. The exchanger operates in steady state; 2. The overall coefficient of heat transfer (U) is assumed constant throughout the exchanger. 3. Heat losses to the surroundings are negligible. 4. The four inlet and outlet temperatures of the heat exchanger streams are known. A complementary approach to analytical solutions is presented by Computational Fluid Dynamics (CFD), which is the science of determining the numerical solution of the governing equations for the fluid flow while advancing the solution through space or time to

obtain a numerical description of the complete flow field of interest. The equations can represent steady or unsteady, compressible or incompressible, and viscous or non-viscous flows, including the behavior of reactive and non-ideal fluids. Likewise,[11], defines it as an established approach to solving fluid dynamics problems by using numerical methods and therefore constitutes an economical alternative to wind and water tunnel testing in addition to the fabrication of other types of prototypes for testing purposes. Furthermore, [6, 12]state that CFD can be likened to a flexible experimental laboratory located on the computer where virtual prototypes are built that easily adapt to a wide variety of physical conditions and allow experimenting with them without building a large-scale physical model or expensive test equipment. In this way, CFD provides an effective platform where various design options can be tested and optimized in a short time and at a relatively low cost. The most important factor in this study is to save a lot of time and money in the experimental study by using CFD.

Currently, CFD is one of the most widely used tools in the design and optimization of heat exchangers due to its flexibility, accuracy, and breadth of application to a wide range of research and engineering problems in many fields of study and industries, including aerodynamics and erosive space analysis, meteorological simulation, natural sciences and environmental engineering, design and analysis of industrial systems, biological engineering and fluid flows, and engine and combustion analysis. Similarly, [13] highlight how CFD analysis helps to analyze the characteristics of a heat exchanger because flow around and within the heat exchanger tubes is very complex, and sophisticated equipment is needed to capture the visual effects. Therefore, CFD analysis allows the observation of flow characteristics at inaccessible locations of shell and tube heat exchangers.

This paper develops the thermal design of a shell and tube heat exchanger by articulating analytical and numerical solutions. The former is performed by the traditional method of logarithmic mean temperature difference (LMTD) and the latter using computational fluid dynamics (CFD) using SolidWorks software and its complement Flow Simulation, which together are solutions concurrent and non-exclusive, thus optimizing the design process of the heat exchanger.

2. Methodology of Heat Exchanger Thermal Design: Analytical Solutions

Based on the thermophysical properties of fluids, the laws of thermodynamics, and heat transfer processes, the main objective of the thermal design is to determine the geometric and functional parameters of the heat exchanger such as number, length, and type of tubes, internal diameter and length of the shell, number, arrangement, and separation of the baffles, number of fluid passages through the tubes, spacing and location of the tubes inside the shell, as well as the materials that will make up each of the parts of the equipment. For this purpose, the analytical thermal design solutions proposed by several authors such as [8, 14, 15, 10, 16] are used, which can be methodologically grouped into the following steps: 1) Initial specifications of the exchanger; 2) Establish the thermophysical properties of the fluids at their average temperatures, find the heat flux of the exchanger and the mass flows of the fluids; 3) Calculate the corrected logarithmic mean temperature difference for the heat exchanger; 4) Review the convective heat transfer coefficients of the fluids flowing through both the tubes and shell of the exchanger; 5) Determine the overall clean and dirty heat transfer coefficients; 6) Determine the heat transfer areas with the clean and dirty transfer coefficients; and estimate the heat exchanger oversize surface, and 7) Selection of appropriate equipment for the required heat exchange. Shell diameter, number of tubes. The data obtained in the analytical solutions are the input to perform the numerical simulation design where the results are validated and possible solutions for the improvement of the heat exchanger performance can be proposed.

2.1 Heat Exchanger Thermal Design Methodology Using Numerical CFD Solutions

The initial step in performing a CFD simulation is to specify an appropriate mathematical model of reality. According to [17], CFD analysis involves three main steps, preprocessing, processing, and post-processing. In the first one, a 3D model of the geometry under study is defined and generated in a proper way by dividing it into elements which is known as meshing, and boundary conditions are applied to it. In processing, the properties of the fluids under study and the physical science model are found; finally, in post-processing, the results are obtained with various techniques such as weight and velocity shape plots, vectors, streamlines, and temperature contouring. These results can be broken down numerically and graphically using either 2-D or 3-D representations.

[12, 18] explain that the CFD method is based on the numerical solution of three main equations: number (1) is the mass conservation equation, also known as the continuity equation; number (2) corresponds to the momentum conservation equation, or Navier-Stokes equation; and number (3) refers to the energy conservation equation. In fact, [5] state that in a rectangular coordinate system the equations for calculating the fluid flow and heat transfer of a heat exchanger can be written as follows:

Conservation of mass equation (Continuity)

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{U}) = 0 \tag{1}$$

Conservation of Momentum Equation (Navier-Stokes equations)

$$\frac{\partial(\rho u)}{\partial t} + \nabla \cdot (\rho u U) = \nabla \cdot (\nabla u \cdot \eta_t) + S_U - \frac{\partial p}{\partial x}$$

$$\frac{\partial(\rho v)}{\partial t} + \nabla \cdot (\rho v U) = \nabla \cdot (\nabla v \cdot \eta_t) + S_v - \frac{\partial p}{\partial y}$$

$$\frac{\partial(\rho w)}{\partial t} + \nabla \cdot (\rho w U) = \nabla \cdot (\nabla w \cdot \eta_t) + S_w - \frac{\partial p}{\partial z}$$
(2)

Conservation of energy equation

$$\frac{\partial(\rho T)}{\partial t} + \nabla \cdot (\rho T U) = \nabla \cdot \left(\frac{\lambda}{C_p} \nabla T \right) + S_T$$
(3)

Where U is the velocity vector; u , v , and w are the velocity components in the x , y , and z directions, respectively; η_t is the coefficient of turbulent viscosity; S is the volumetric heat source term; S_U , S_v , S_w are source terms for the x , y and z directions, respectively; λ is the heat conductivity; C_p is the specific heat of the fluid.

3. Problem Definition

Thermal design of a shell and tube heat exchanger is required to cool methanol from 48°C to 34°C using water that is available at 13°C and can be heated up to 25°C. Methanol enters the exchanger at a rate of 8300 kg/hr.

Figure 1 shows the main parts of the heat exchanger as well as the inlet and outlet temperatures of the hot and cold fluids along the length of the heat exchanger.

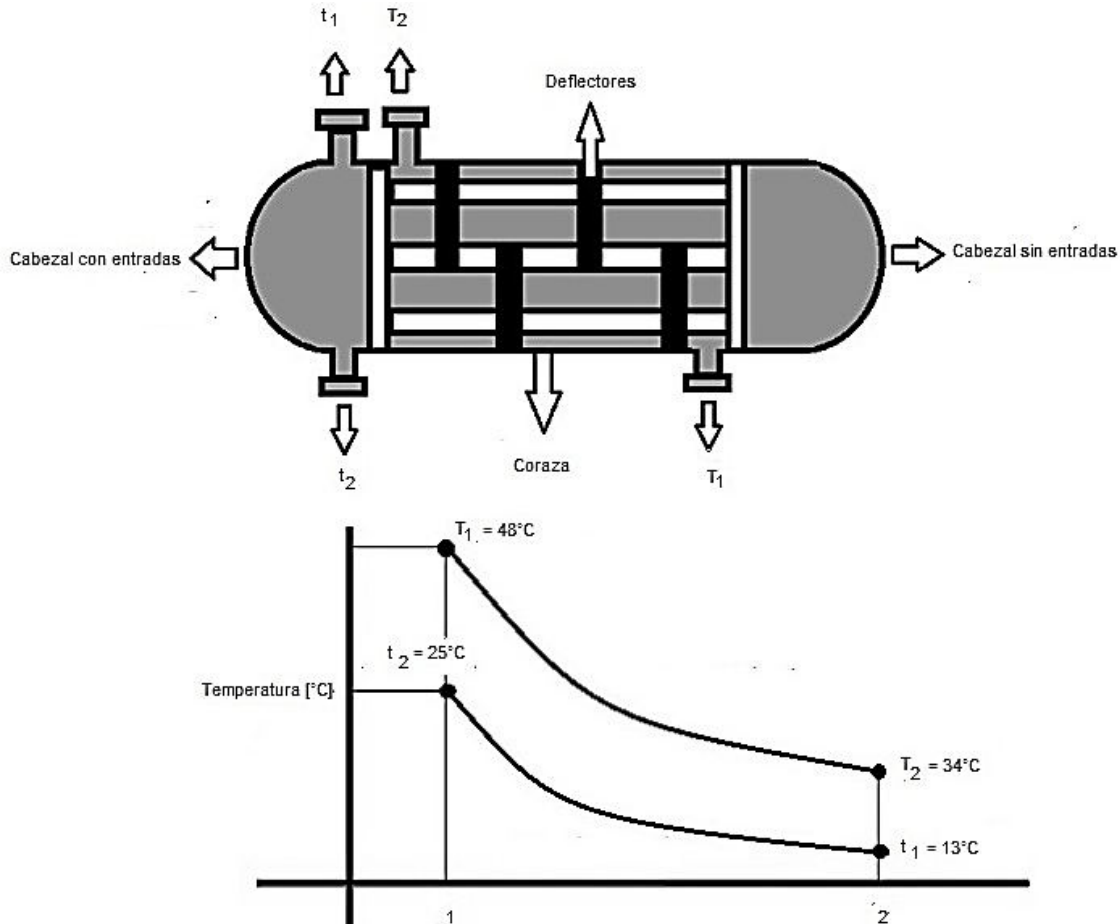


Figure 1. Main parts of the heat exchanger and temperature variation diagram of the hot and cold fluids inside the heat exchanger.

4. Results Analytical Solutions

The analytical solutions are found considering the following assumptions: Heat losses to the surroundings are negligible, the properties of the working fluids are considered constant inside the heat exchanger, kinetic and potential

energy changes are negligible, and the flow conditions are fully developed in both fluids. Thus, the described methodology is applied to find the analytical solutions for the thermal design of the heat exchanger.

4.1 Initial heat exchanger specifications

For the thermal design, methanol is considered as the hot fluid, and water is considered as the cold one and the latter will flow through the tubes since it is the most corrosive fluid, methanol will move on the shell side [10]. Table 1 shows the initial design conditions of the exchanger referred to in the definition of the problem.

Table 1. Initial design parameters of the heat exchanger.

	Hot fluid	Cold fluid
Fluid	Methanol	Water
Inlet temperature	48°C	13°C
Outlet temperature	34°C	25°C
Mass flow rate	8300 kg/hr	-----

On the other hand, 3/4" BWG 14 tubes of three meters in length are selected, whose geometric characteristics are listed and shown in Table 2. These tubes will be arranged in a square arrangement since the cleaning of the outer surfaces of the tubes will be required, following the design guidelines proposed by [8, 10], the tube pitch (PT) is specified as 0.0254 m as shown in Figure 2. Finally, it is decided that the exchanger will have two passages through the tubes and one in the shell.

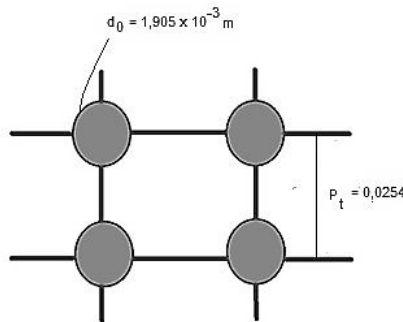


Figure 2. Geometry of square tube arrangement.

Table 2. Characteristics of the tubes selected for the heat exchanger.

Geometric characteristics of the tube 3/4 BWG 14		
d _o	d _i [m]	e [m]
1,905x 10 ⁻³ m	0,01483 m	2,1082 x 10 ⁻³ m

Establish the thermophysical properties of the fluids at their average temperatures and find the heat flux of the heat exchanger and the mass flow rates of the fluids.

Equations 4 show the results of the average temperatures of methanol and water, based on their inlet and outlet temperatures of the exchanger.

$$\bar{T} = \frac{T_1 + T_2}{2} = \frac{48 + 34}{2} = 41^\circ\text{C}$$

$$\bar{t} = \frac{t_1 + t_2}{2} = \frac{13 + 25}{2} = 19^\circ\text{C}$$
(4)

At the average temperatures of the fluids listed in Equation (4), the thermophysical properties of each fluid are established as shown in Table 3.

Table 3. Thermophysical properties of methanol and water at intermediate temperatures.

Property	Units	Methanol (41°C)	Water (19°C)
Density	Kg/m ³	772,08	998,4
Dynamic viscosity	Pas	4,38 x 10 ⁻⁴	1,14 x 10 ⁻³
Specific heat	J/Kg °K	2615,5	4185,28
Thermal conductivity	W/°Km	0,184	0,59

With the information corresponding to methanol and using Equation (5), the heat flow in the exchanger is determined.

$$q = \dot{m}_c C_{pc} (T_1 - T_2) = (2,30)(2615,15)(48-34) = 84411,23 \text{ W} \quad (5)$$

The mass flow rate of water required to satisfy the thermal demand of the heat exchange process is then determined using Equation (6).

$$\dot{m}_i = \frac{q}{C_{pf}(t_2 - t_1)} = \frac{84411,23}{(4185,28)(25-13)} = 1,68 \frac{\text{Kg}}{\text{s}} \quad (6)$$

4.2 Calculate the corrected logarithmic mean temperature difference for the heat exchanger.

Since the inlet and outlet temperatures of methanol and water are known, the logarithmic average temperature of the heat exchanger is calculated by means of Equation (7).

$$\Delta_{LMTD} = \frac{(T_1 - t_2) - (T_2 - t_1)}{\ln \left(\frac{T_1 - t_2}{T_2 - t_1} \right)} = \frac{(48-25) - (34-13)}{\ln \left(\frac{48-25}{34-13} \right)} = 21,98^\circ\text{C} \quad (7)$$

Since the heat exchanger to be designed has two passes through the tubes and one in the shell, the logarithmic mean temperature must be corrected using the F factor calculated using Equations (8), (9), (10), (11), and (12).

$$R = \frac{T_1 - T_2}{t_2 - t_1} = \frac{48 - 34}{25 - 13} = 1,16 \quad (8)$$

$$P = \frac{t_2 - t_1}{T_1 - t_1} = \frac{25 - 13}{48 - 13} = 0,34 \quad (9)$$

$$\alpha = \left(\frac{1 - RP}{1 - P} \right)^{\frac{1}{N}} = \left(\frac{1 - 0,39}{1 - 0,34} \right)^{\frac{1}{2}} = 0,95 \quad (10)$$

$$s = \frac{\alpha - 1}{\alpha - R} = \frac{0,95 - 1}{0,94 - 1,16} = 0,23 \quad (11)$$

$$F = \frac{\sqrt{R^2 + 1} \ln \left(\frac{1 - s}{1 - RS} \right)}{(R - 1) \ln \left[\frac{2 - s(R + 1 - \sqrt{R^2 + 1})}{2 - s(R + 1 + \sqrt{R^2 + 1})} \right]} = \frac{(1,53) \ln(1,05)}{(0,16) \ln \left(\frac{1,85}{1,15} \right)} = 0,99 \quad (12)$$

The correction factor F obtained is multiplied by the logarithmic mean temperature and thus its corrected value is obtained, as shown in Equation (13).

$$\Delta_{LMTDC} = (\Delta_{LMTD})(F) = (21,98)(0,99) = 21,76^\circ\text{C} \quad (13)$$

Review of the convective heat transfer coefficients of the fluids flowing through both the tubes and the shell of the heat exchanger.

Authors such as [14] relate the film convective heat transfer coefficients to the following values: methanol: $h_0 = 900 \text{ W/m}^2\text{°C}$, water: $h_i = 4000 \text{ W/m}^2 \cdot \text{°C}$

4.3 Determination of the overall clean and dirty heat transfer coefficients.

When the heat exchanger is new and the heat transfer surfaces are clean, its overall heat transfer coefficient is calculated by Equation (14); however, with most liquids, a film of dirt or scale will build up on the heat transfer surfaces over a period of time. This process is called fouling and results in decreased heat exchanger performance due to the added thermal resistance of the fouling films. Fouling is considered by means of empirically determined fouling factors, R_f and R_c , representing the thermal resistances of the dirt films inside and outside the tube, according to [14, 10] for water $R_f = 0,0002 \text{ m}^2 \cdot \text{°K/W}$ and for methanol $R_c = 0,000352 \text{ m}^2 \cdot \text{°K/W}$. In such a situation, the overall heat transfer coefficient is calculated by Equation (15).

$$U = \left[\frac{1}{h_o} + \frac{r_o}{r_i} \frac{1}{h_i} + \left(r_o \frac{\ln\left(\frac{r_o}{r_i}\right)}{k} \right) \right]^{-1} = 598,93 \frac{W}{m^2 \cdot K} \quad (14)$$

$$U_E = \left[\frac{1}{h_o} + \frac{r_o}{r_i} \frac{1}{h_i} + R_f + R_c + \left(r_o \frac{\ln\left(\frac{r_o}{r_i}\right)}{k} \right) \right]^{-1} = 497,6 \frac{W}{m^2 \cdot K} \quad (15)$$

Determine the heat transfer areas with the dirty and clean transfer coefficients and estimate the heat exchanger oversize surface area.

In this step, initially, the clean and dirty heat transfer areas of the heat exchanger are calculated by means of Equations (16) and (17)[14]. Then, the oversize surface is determined using Equation (18). In this case, it is 20.21%, which is within the parameters established by [8].

$$A = \frac{q}{U \Delta_{LMTDC}} = \frac{(84411,23)}{(598,93)(21,76)} = 6,48 m^2 \quad (16)$$

$$A_E = \frac{q}{U \Delta_{LMTDC}} = \frac{(84411,23)}{(497,6)(21,76)} = 7,79 m^2 \quad (17)$$

$$SSD = \left(\frac{A_E}{A} - 1 \right) \times 100 = 20,21\% \quad (18)$$

4.4 Selection of appropriate equipment for the heat exchange required

The abbreviation PR is known as the constant pitch of the tubes and is obtained by means of Equation (19). In addition, it is recommended that this value is between 1.25 and 1.5, where Pt corresponds to the pitch of the tubes and the inner diameter of the tubes. From Equation (20), the inner diameter of the shell of the heat exchanger is determined where CTP is called the tube count constant, for exchangers with two pitches in the tubes and one in the shell it has a value of 0.90; CL is known as the geometrical arrangement constant of the tubes and has a value of 1 for configurations of 45° and 90°[14].

$$PR = \frac{P_t}{d_o} = \frac{0,0254}{1,905 \times 10^{-2}} = 1,33 \quad (19)$$

$$D_s = 0,637 \sqrt{\frac{CTP}{CL} \left[\frac{A_E (PR)^2 d_o}{L} \right]^{\frac{1}{2}}} = 0,637 \sqrt{\frac{0,90}{1} \left[\frac{(7,79)(1,33)^2 (1,905 \times 10^{-2})}{3} \right]^{\frac{1}{2}}} = 0,19 \text{ m} \quad (20)$$

Next, the number of tubes required in the heat exchanger is determined using Equation (21).

$$N_t = 0,785 \left(\frac{CTP}{CL} \right) \frac{D_s^2}{(PR)^2 (d_o)^2} = 0,785 \left(\frac{0,90}{1} \right) \frac{(0,19)^2}{(1,33)^2 (1,905 \times 10^{-2})^2} = 39,73 = 40 \quad (21)$$

For [14], the spacing between baffles should be done very carefully, and thus it is proposed that an optimal spacing between them should be between 0.4 and 0.6 times the inner diameter of the shell. In the same way, this author recommends using baffles with a cut-off between 25% and 35% of the inner diameter of the shell, as in Equation (22).

$$B = 0,6 D_s = (0,6)(0,19) = 0,114 \text{ m} \quad (22)$$

Finally, the velocity of the water inside the tubes is calculated by means of the Equation (23).

$$v = \frac{\dot{m}}{\rho A} = \frac{(1,68)}{(998,4)(1,96 \times 10^{-3})} = 0,85 \frac{m}{s} \quad (23)$$

5.CFD Simulation Results

CFD analysis in SolidWorks and its Flow simulation add-on will provide a visualization with simulated illustrations of heat exchanger temperature and flow velocity conditions.

5.1 Procedure

Initially, a CAD model of the heat exchanger was developed using Solidworks 2020 software, as shown in Figure 3, using the information in Table 4, which corresponds to the geometric dimensions and materials of the different parts of the heat exchanger obtained from the analytical solutions.

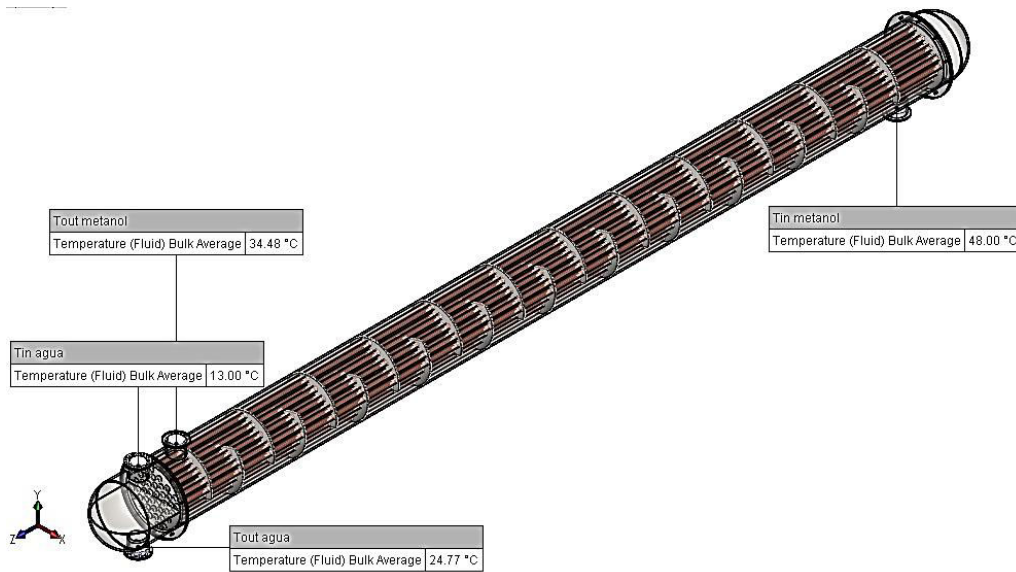


Figure 3. CAD model prepared with the information from the analytical solutions.

Table 4. Geometric dimensions and materials of the heat exchanger found with the analytical solutions.

Description	Units	Value
Length of shell and tubes	m	3
Inner diameter of the shell	m	0,19
Number of passages through tubes		2
Number of tubes		40
Outer diameter of tubes	m	$1,905 \times 10^{-2}$
Inner diameter of tubes	m	$1,4 \times 10^{-2}$
Number of baffles (20%)		23
Spacing between baffles	m	0,12
Distance between tube centers	m	0,0254
Tube type		$\frac{3}{4}$ BWG 14
Tube center-to-center distance		0,0254
Shell, headers, baffles material		Acero AIS304
Tube material		Cobre
Tube arrangement		Cuadrado

Subsequently, the CAD model was discretized with a level 5 mesh, which used hexahedra with a total of 1050084 elements, of which 619728 correspond to fluid cells, 430856 correspond to solid elements and 324070 to fluids connected to solids, part of this mesh is shown in Figure 4.

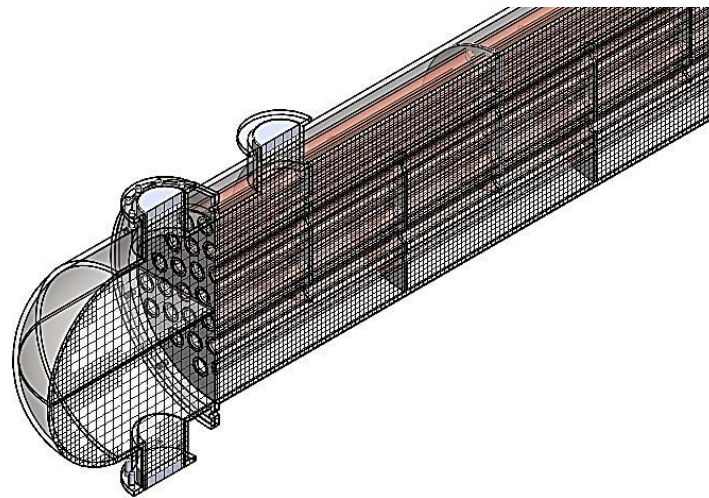


Figure 4. Mesh of the CAD model channel exchanger.

Finally, the boundary conditions of the heat exchanger are applied, corresponding to the inlet temperatures and mass flows of the hot and cold fluids, as well as their outlet pressures. Table 5 shows this information.

Table 5. Boundary conditions applied to the heat exchanger.

Exchanger boundary conditions		
	Hot fluid	Cold fluid
Fluid	Methanol	Water
Inlet temperature	48°C	13°C
Outlet pressure		
Mass flow rate	2,30 Kg/s	1,68 Kg/s

5.2 Processing

The SolidWorks flow simulation plug-in was used to monitor the water and methanol working fluid flows through the virtual model, using the Wizard wizard to configure the working parameters of the heat exchanger such as internal flow analysis including conduction in solids, laminar and turbulent flow, adiabatic process in the heat exchanger walls, etc.

5.3 Postprocessing

SOLIDWORKS software and its Flow Simulation add-on use CFD analysis to simulate the flow inside the heat exchanger to provide a visualization of the temperature gradient and velocity of the hot and cold fluids along the heat exchanger. The flow simulation results for this study include shear plots, surface plots and flow paths. Temperature and fluid velocity are the main parameters studied. The shear plot shows the findings in a specific area, while the flow paths show the fluid flow along the exchanger or in a given area. Finally, the transmission of parameters to the surface is shown in the surface plot, its main use is to analyze the temperature transfer to the outer and inner surface in the solid domain of the exchanger.

5.4 CFD Analysis: Cut charts

Cut charts are used to examine attributes from a section perspective. This ultimately allows the analysis to be performed on a cross-sectional region and the plotting of maximum and minimum parameter values. It also provides an overview of attributes the fluid velocity and temperature as they flow through the exchanger. Figure 5 shows the temperature distribution inside the heat exchanger in a side section view, showing the temperature change of the water and methanol from the time they enter the heat exchanger until they leave it, the water enters the heat exchanger at 13°C and leaves at 24.77°C, while the methanol enters at 48°C and leaves at 34.48°C.

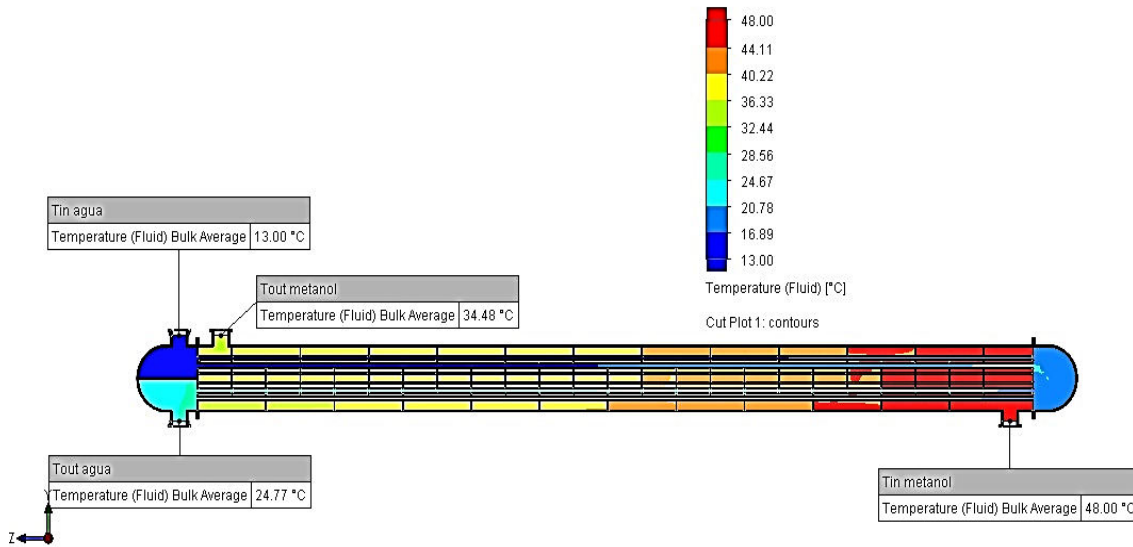


Figure 5. Cut-off graph of working fluid temperatures inside the heat exchanger.

In the same way, Figure 6 shows the heat exchanger cutoff graph where the fluid velocity vector field is represented with its corresponding maximum and minimum temperatures.

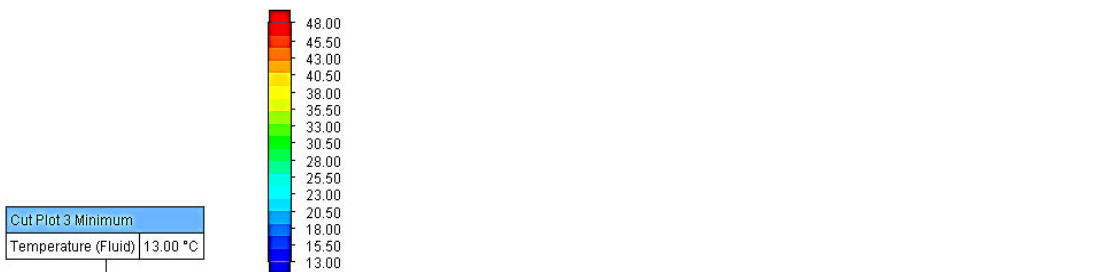


Figure 6. Vector field shear plot of velocities of the working fluids inside the heat exchanger.

Finally, Figure 7 shows the flow lines of the fluids inside the heat exchanger.

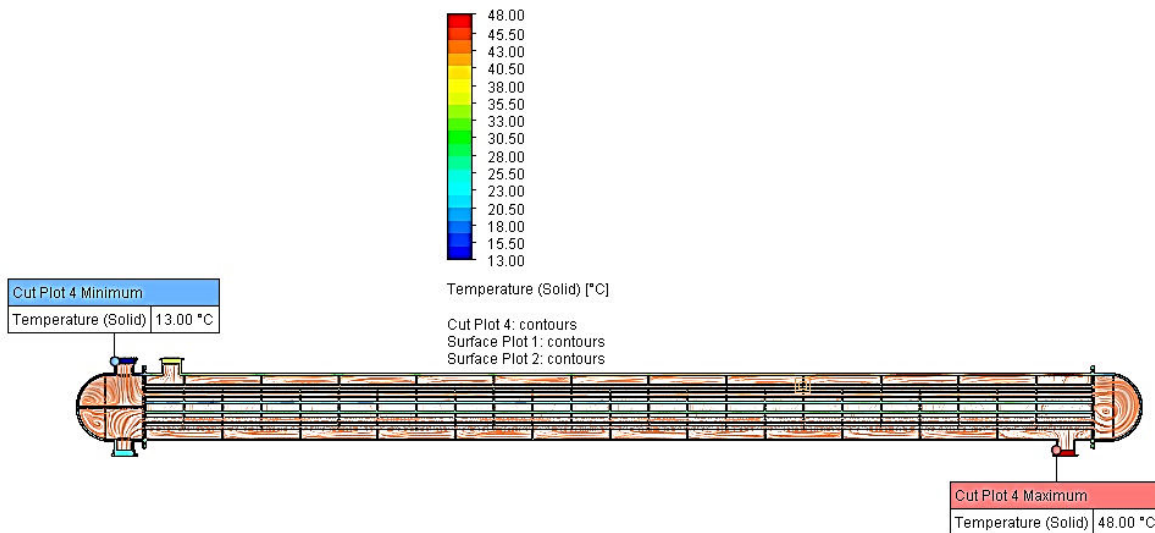


Figure 7. Cutaway plot showing the streamlines of the working fluids inside the heat exchanger.

5.5 CFD Analysis: Surface Plots

Surface graphs are another type of analysis that provides a clearer visual representation of the variations in the attributes under study. Thus, Figure 8 shows the temperature variation at the exchanger shell surface as well as the

corresponding maximum (48 °C) and minimum (13°C) temperature points. The temperature is high near the methanol inlet and gradually decreases towards the water inlet.

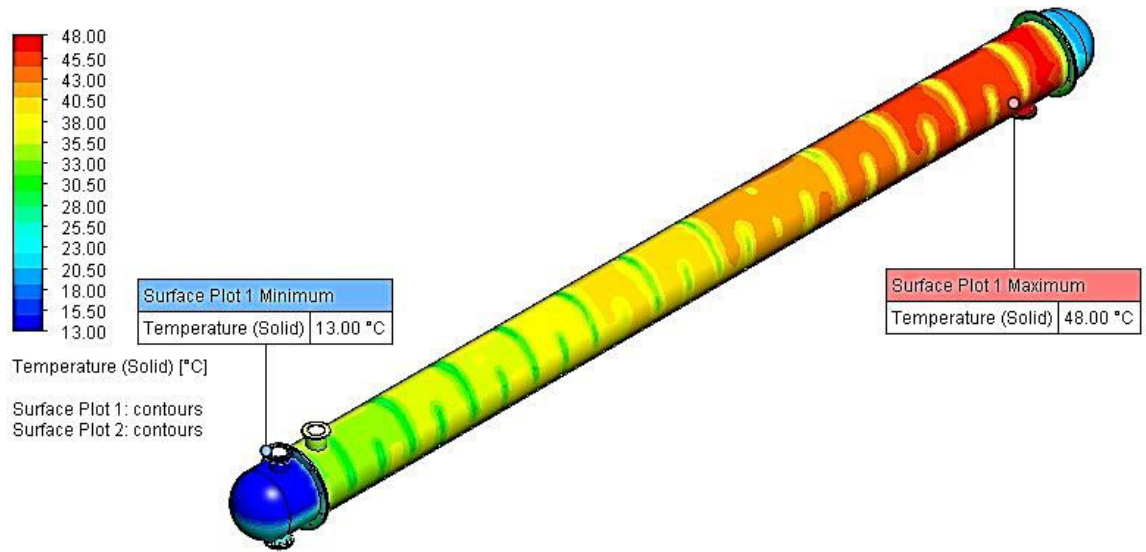


Figure 8. Surface temperature graph on the shell Surface.

Figure 9 shows the temperature distribution inside the heat exchanger in the solid domain of the heat exchanger (tubes, separators, and baffles), where the maximum temperature points are 42.21°C and the minimum temperature is 15.24°C.

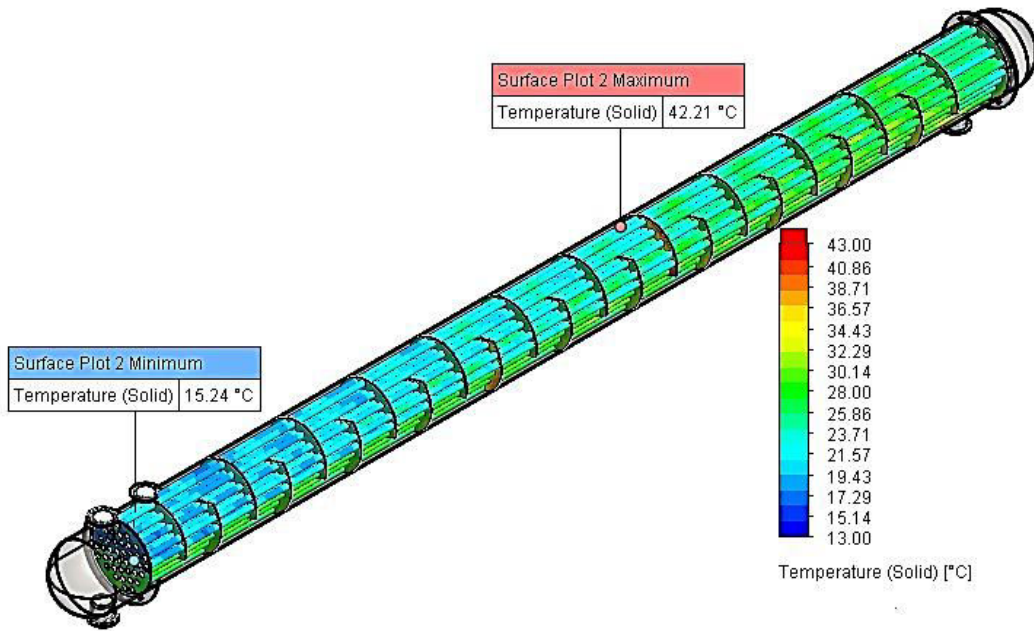


Figure 9. Heat exchanger solid domain surface temperature graph.

5.6 CFD Analysis: flow paths

The flow paths show how the fluids move through the shell and tubes in the heat exchanger. Figure 10 shows how the temperature of the water varies as it moves through the tubes, where the inlet water temperature is 13°C, a mass flow rate of 1.68 kg/s and an outlet temperature of 24.77°C can be observed.

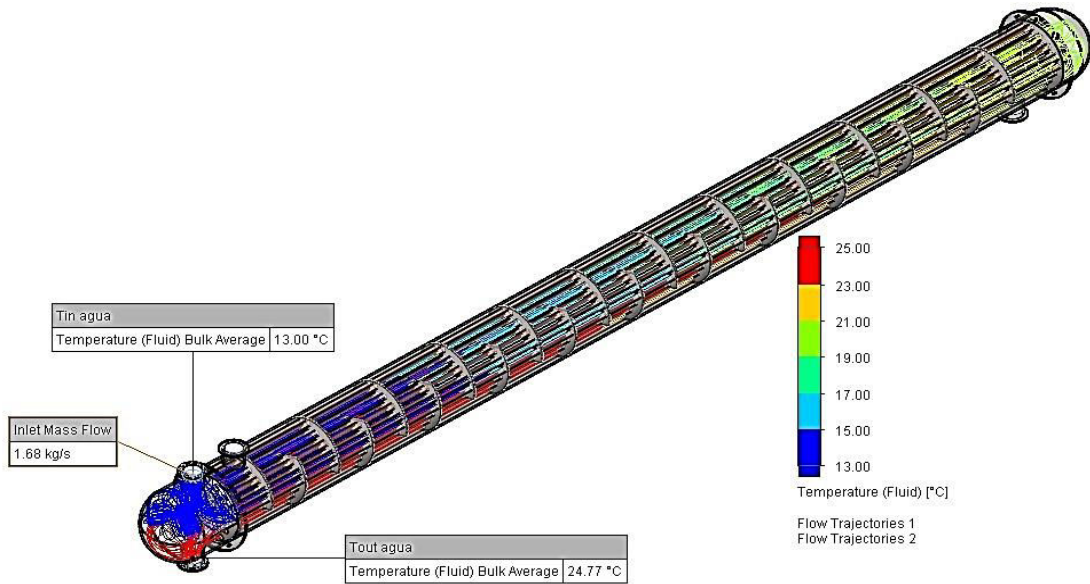


Figure 10. Water flow path inside the heat exchanger tubes.

Similarly, Figure 11 shows the trajectory of methanol inside the exchanger, where it enters through the lower side of the shell with a mass flow rate of 2.3 kg/s and a temperature of 48°C and exits at 34.48°C through the upper side of the shell.

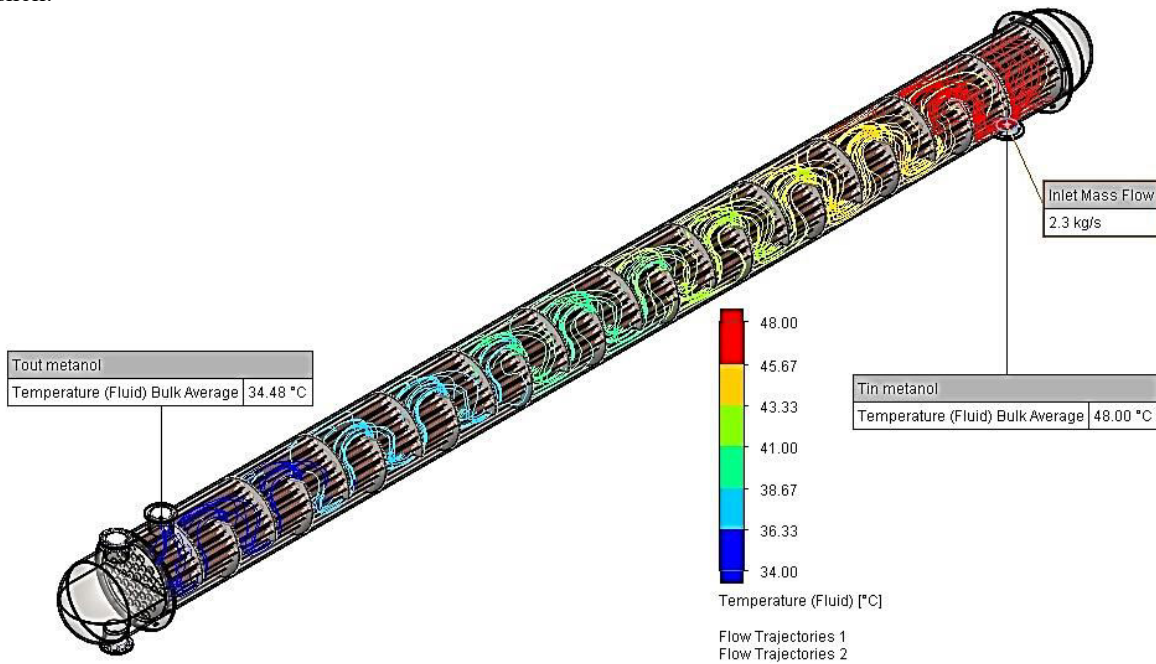


Figure 11. Methanol flow path inside the heat exchanger Shell.

Figure 12 shows the simulation of the trajectories of the two fluids, with their corresponding inlet and outlet temperatures.

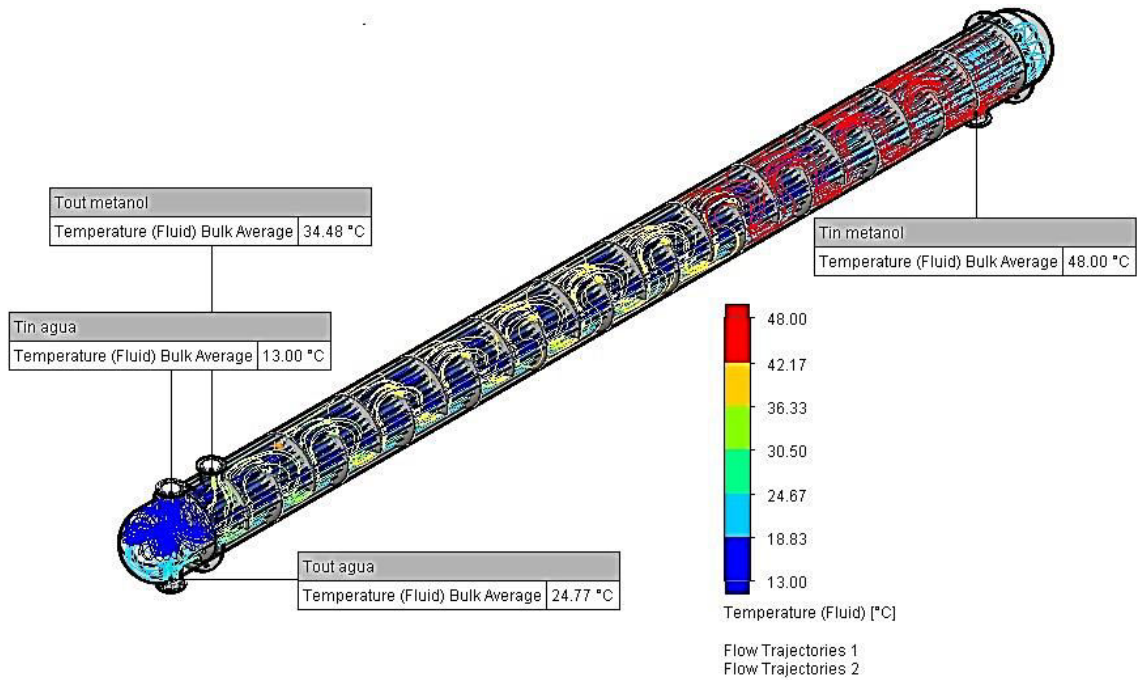


Figure 12. Flow paths of the two fluids inside the exchanger.

Finally, Figure 13 shows the velocity gradient of the water inside the tubes from the inlet to the outlet of the heat exchanger.

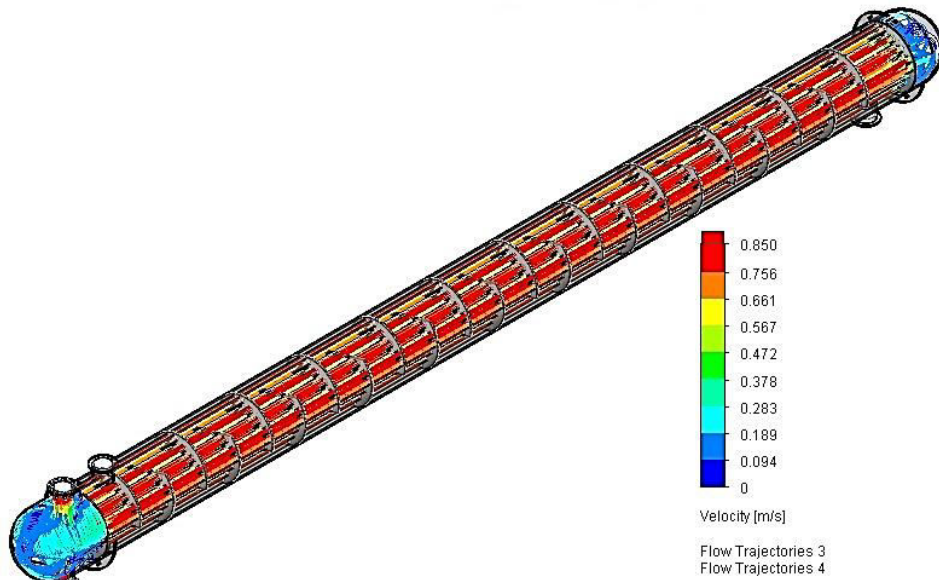


Figure 13. Velocity gradient of the water inside the tubes.

6. Validation of the CFD Model

Table 6 shows the percentage differences between the initial design conditions of the heat exchanger and those of the numerical simulation for the fluid outlet temperatures and the water velocity inside the tubes.

Table 6. Percentage difference between design values and analytical solutions.

	Initial Conditions	Numerical solutions	Percentage difference
Methanol outlet temperature °C	34	34.48	1.41%
Water outlet temperature	25	24.77	0.92%
Water velocity in the pipes [m/s].	0.85	0.85	

NOMENCLATURE		
T_1	Inlet temperature of the hot fluid.	$^{\circ}\text{C}$
T_2	Hot fluid outlet temperature.	$^{\circ}\text{C}$
\bar{T}	Average hot fluid temperature	$^{\circ}\text{C}$
t_1	Cold fluid inlet temperature	$^{\circ}\text{C}$
t_2	Cold fluid outlet temperature	$^{\circ}\text{C}$
\dot{m}_c	Hot fluid mass flow rate	Kg/s
C_{pc}	Specific heat of hot fluid	$\text{J/Kg } ^{\circ}\text{K}$
C_{pf}	Specific heat of cold fluid	$\text{J/Kg } ^{\circ}\text{K}$
Δ_{LMTD}	Logarithmic mean temperature	$^{\circ}\text{C}$
F	Temperature correction factor	
$\frac{r_o}{d_o}$	Outer tube radius/outer tube diameter	m
$\frac{r_i}{d_i}$	Tube inner radius/tube inner diameter	m
e	Tube wall thickness	m
h_o	Assumed film heat transfer coefficient of the fluid flowing through the shell	$\text{W/m}^2\text{ } ^{\circ}\text{K}$
h_i	shell	$\text{W/m}^2\text{ } ^{\circ}\text{K}$
k	Assumed pellicular heat transfer coefficient of the fluid flowing through the tubes	
A_E	Thermal conductivity	m^2
A	Heat transfer area under fouling conditions	m^2
L	Heat transfer area	m
n	Total length of tubes	
N_d	Number of passages through the tubes	
R_c	Number of baffles	$\text{m}^2\text{ } ^{\circ}\text{K/W}$
R_f	Fouling coefficient of the hot fluid	$\text{m}^2\text{ } ^{\circ}\text{K/W}$
q	Fouling coefficient of cold fluid	W
SSD	Heat flux	
B	Oversize surface	m
ρ	Baffle spacing	Kg/m^3
U	Fluid density	
D_s	Overall heat transfer coefficient	m
N_t	Shell diameter	
U_E	Number of heat exchanger tubes	
P_t	Overall heat transfer coefficient under fouling conditions	m
v	Tube pitch	m/s
PR	Velocity of water in the tubes	
CTP	Pitch constant of tubes	
CL	Tube count constant	

7. Conclusions

The CDF numerical simulation allows digital modeling to validate the analytical solutions, these tools allow to optimize time and costs before proceeding to the construction of the exchanger.

The engineering solutions are neither exclusive nor independent, but complementary, so it is possible to articulate analytical and numerical solutions to optimize the design of the exchanger.

Author Contributions: Conceptualization J.C., methodology J.C., writing—original J.C., J.U., draft preparation J.U., writing-review J.U., and editing were carried out by J.U. The authors have read and agreed to the published version of the manuscript.

Funding: This research received no external funding.

Data Availability Statement: The data used in the study shall be made available to other researchers by request.

Acknowledgments: The authors thank Universidad Cooperativa de Colombia for their support in completing this project.

Informed Consent Statement: Informed consent was obtained from the subjects involved in the study.

Conflicts of Interest: The authors declare no conflicts of interest.

References

- [1] Yadav, G.; Janugade S.; Patil M. Numerical Simulation of Shell and Tube Heat Exchanger by using CFD, *IARJSET* **2017**,4(1), 105–110. <https://doi.org/10.17148/iarjset/ncdmete.2017.25>
- [2] Saffarian, M.; Fazelpour F.; Sham, M. Numerical study of shell and tube heat exchanger with different cross-section tubes and combined tubes, *International Journal of Energy and Environmental Engineering* **2019**, 10, 33-46. <https://doi.org/10.1007/s40095-019-0297-9>
- [3] Pal, E.; Kumar, I.; Joshi J.; Maheshwari, N. CFD simulations of shell-side flow in a shell-and-tube type heat exchanger with and without baffles, *Chemical engineering science* **2016**, 143, 314-340. <https://doi.org/10.1016/j.ces.2016.01.011>
- [4] Kishan, R.;Singh D.;Sharma, A. CFD analysis of heat exchanger models design using ansys fluent, **2020**. <https://doi.org/10.31224/osf.io/drn4>
- [5] Yang, X.;Zhang, J.;Yu, J.;Wu , Y.;Luo Y.;Guo, J.Numerical simulation on flow and heat transfer at shell-side of flat-tube heat exchangers, *Wuhan University Journal of Natural Sciences***2010**, 15, 427-432. <https://doi.org/10.1115/ht2007-32914>
- [6] Patel P.;Amitesh, P. Thermal Analysis Of Tubular Heat Exchanger By Using Ansys, *Proc. IJERT***2012**, 1(8). <https://www.ijert.org/research/thermal-analysis-of-tubular-heat-exchanger-by-using-ansys-IJERTV1IIS8212.pdf>
- [7] Babu C.;Gugulothu, S. CFD analysis of heat transfer enhancement by using passive technique in heat exchanger, *Int. J. Recent Adv. Mech. Eng.* **2015**, 4, 99-111. <https://doi.org/10.14810/ijmech.2015.4308>
- [8] Kern, S. Operaciones de Transferencia de Calor, 32 ed., México: CECSA, **2001**. <https://es.slideshare.net/slideshow/procesosdetransferenciadecalorkern31edpdf/257545100>
- [9] Carrero Mantilla, J. Equipos de transferencia de calor: una guía de estudio, Departamento de Ingeniería Química, **2011**.<https://repositorio.unal.edu.co/handle/unal/55684>
- [10] Serth R.; Lestina, T. Process heat transfer: Principles, applications and rules of thumb, *Academic press*, **2014**. https://www.researchgate.net/publication/286214207_Process_Heat_Transfer_Principles_Applications_and_Rules_of_Thumb_Second_Edition
- [11] Abeykoon, C. Compact heat exchangers–Design and optimization with CFD, *International Journal of Heat and Mass Transfer***2020**, 146. <https://doi.org/10.1016/j.ijheatmasstransfer.2019.118766>
- [12] Hasgul C.; Cakmak, G. Heat transfer analysis of double tube heat exchanger with wavy inner tube, *hermal Science***2022**, 26(4), 3455-3462. <https://doi.org/10.1016/j.ijheatmasstransfer.2019.118766>
- [13] Khan, A.;Shah, I.;Gul, W.;Khan, T.;Ali Y.;Masood, S. Numerical and Experimental Analysis of Shell and Tube Heat Exchanger with Round and Hexagonal Tubes, *Energies***2023**,16(2), 880. <https://doi.org/10.3390/en16020880>
- [14] Kakac, S.;Liu H.;Pramuanjaroenkij, A. Heat exchangers: selection, rating, and thermal design, **2002**. <https://doi.org/10.1201/9780429469862-9>
- [15] Cao, E. Heat transfer in process engineering, McGraw-Hill Education, **2010**. <https://www.accessengineeringlibrary.com/content/book/9780071624084>
- [16] Nitsche M.;Gbadamosi, R. Heat exchanger design guide: a practical guide for planning, selecting and designing of shell and tube exchangers, **2015**. <https://doi.org/10.1016/b978-0-12-803764-5.00001-8>
- [17] Katarki S.; Malipatil, A. CFD Analysis of Shell and Tube Heat Exchanger for Heat Transfer Capabilities, *International Journal of Engineering and Techniques***2017**, 3(6). <http://www.ijetjournal.org/Volume3/Issue6/IJET-V3I6P15.pdf>

- [18] Wang, D.; Wang, H.; Xing J.; Wang, Y. Investigation of the thermal-hydraulic characteristics in the shell side of heat exchanger with quatrefoil perforated plate, *International Journal of Thermal Sciences* **2021**, 159. <https://doi.org/10.1016/j.ijthermalsci.2020.106580>