

Design of a Double Pipe Heat Exchanger in Counter Flow Configuration Using Numerical Analysis

*Kenny Pérez Capó
Master of Engineering in Mechanical Engineering
Bernardo Restrepo-Torres, PhD
Mechanical Engineering
Polytechnic University of Puerto Rico*

Abstract – *The Double Pipe Heat Exchanger in counter flow configuration consists of one fluids flow from the inner pipe while the other flows in the opposite direction from the outer pipe. This project analyzed and designed this heat exchanger utilizing numerical simulation in ANSYS and facilitated optimization by performing a change in the inner tube material. Based on the second simulation analysis the minimum percentage difference was reduced from 2.70% to 0.70% When changing the inner tube material to copper and maintaining the same initials conditions a 12.07 K decrease in temperature difference was achieved for the hot water. It is found that the visualization of the distribution of the temperature and pressure that numerical simulation provides, gives a better understanding of the heat exchanger and having the system model in ANSYS useful for optimization.*

Key Terms — *ANSYS, Heat Transfer, Optimization and Simulation.*

INTRODUCTION

Description

The purpose of this project design is to analyze and design this heat exchanger in counter flow configuration utilizing numerical simulation in ANSYS and facilitate optimization by performing a change in the inner tube material. Double pipe heat exchangers are the simplest heat transfer devices; one fluid flows through a smaller pipe while the other fluids flow through the annular space between the two pipes. Heat is transferred by conduction and convection. Basically, there are two methods to analyze heat exchangers, one of them is called the Logarithmic Mean Temperature Difference (LMTD) and the other one is the Number of Transfer Units (NTU). Nowadays, manufacturer

companies utilize codes such as ASME Boiler Pressure Vessel and Tubular Exchanger Manufacturers Association (TEMA) to design their heat exchangers although this is a standard method, while utilizing numerical simulation analysis provides insurance that the system will perform as specified and it is not commonly used in practice.

Objectives

The main objectives that this project design intends to provide are:

- Determine pressure distribution in the pipes.
- Check consistency with mathematical model and check numerical errors
- Understand how numerical simulation works

Contributions

The main contributions that this project design intends to provide are:

- Develop physical intuition about this flow through visualization results.
- Determine the heat transfer characteristics of a double pipe heat exchanger.
- Establish determining factors that induced the difference in theoretical and numerical simulation results.

THEORETICAL BACKGROUND

Heat transfer is the study of thermal energy transport that occurs between states of matter when a difference in temperature exists and only in the direction from the hot object to a cold one. This can occur in three ways: conduction, convection, and radiation. Conduction is the transfer of energy cause by physical molecular interaction. Some examples are using a heating pad to warm your muscles, holding a cube of ice in your hands, and a heat exchanger when the heat of the hot fluid is

transferred to a cooler fluid through the walls of the pipes. This heat transfer can be calculated using Fourier's Law, which includes the thermal conductivity k of the material. The thermal conductivity is an indicator of how effective the material transfer heat. Convection is the transfer of energy due to streams of moving fluids. There are two types of convection, free or natural convection. Natural convection is when a fluid moves without using external energy and forced convection that utilize mechanical means to move the fluid such as a fan. Examples of applied convection mechanisms includes boiling of water and the radiator of a car. This heat transfer can be calculated using Newton's Law of Cooling which includes the convective heat transfer coefficient h , this coefficient depends on the characteristic of the system, such as velocity and properties of the fluid and geometry of the medium. Radiation is the transfer of energy in form of electromagnetic waves consequence of vibrational and rotational motions of molecules, atoms, and electrons of a substance. Energy transfer by radiation is fastest and it suffer no limitations in vacuum. This heat transfer can be categorized in blackbody that only depends on Stefan-Boltzmann constant and gray body where in addition to the Stefan-Boltzmann constant it depends in the emissivity coefficient.

A commonly used device to transfer heat in the industry, widely used in refrigeration, power plants, and chemical plant are heat exchangers [1]. A heat exchanger is any apparatus in which heat is transferred between a warmer and colder fluids. There is various type of heat exchangers like shell and tube, plates, and double pipe. Double pipe heat exchangers are the simplest heat transfer devices, one fluid flows through a smaller pipe while the other fluids flow through the annular space between the two pipes. This type of heat exchanger can be configured to be in parallel flow, crossflow, or counter flow where crossflow can have mixed or unmixed. On this system counter flow configuration is always more favorable than the parallel current [2]. In this type of heat exchanger heat is transferred by conduction and convection,

the two methods commonly used to analyze this are the Logarithmic Mean Temperature Difference (LTMD) and the Number of Transfer Units (NTU). Prior studies mention that double pipe heat exchangers performance is analyzed using some parameters, including Nusselt Number (Nu), Reynolds Number (Re), friction factor (f), and effectiveness (ϵ) [3]. Another important parameter is fouling which refers to undesired accumulation of solid material on heat exchangers surfaces but due to the simplicity of the proposed system this will not be taken into consideration.

A useful tool to model a system of this magnitude is numerical simulation. In this project design ANSYS will be used to perform this numerical simulation. ANSYS is a software package that consists of tools where 3D designs and simulations of real-world applications can be modeled. ANSYS utilizes the fundamental laws of fluid flow that are conservation of mass, conservation of momentum and conservation of energy and solves them utilizing the Finite-Volume Method (FVM). FVM is a method that divides the domain into multiple control volumes or cells also called mesh. Then the integral forms of the conservation's laws are discretized. When applying the discretization, into the integral conservations laws we get algebraic equations and an error then we linearized these algebraic equations, and we get another small error called Linearization. Finally, the programs solve iteratively updating guess and stops after the imbalances are below tolerances this rigorous defined process can be seen in Figure 1. An important thing to keep in mind that to lower the discretization error one refines the mesh and to lower the linearization error iterate until the imbalance tolerance converges. This project design will design a preliminary double pipe heat exchanger in counter flow to be modeled utilizing ANSYS and with those results the design if needed will change.

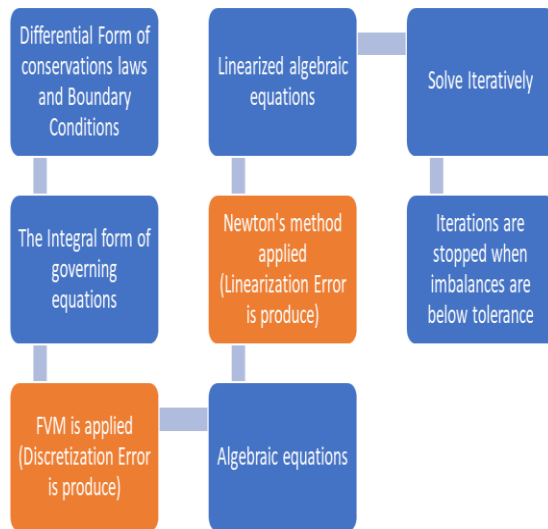


Figure 1
Process of the Conservation's Laws are Resolve Using the Integral Form in Numerical Simulation

METHODOLOGY

The first step is to design the geometry of the double pipe heat exchanger establishing parameters including but not limited to the materials of the pipes, selection of fluid, and size of the pipes. As mentioned above the LTMD method will be used to analyze the heat exchanger and to determine the characteristics that influenced the heat transfer in a double pipe heat exchanger. Then, a numerical simulation will be performed in ANSYS following these steps:

- Geometry Preparation
- Constructing the Mesh
- Configuring the Mathematical Model
- Setting up and running the Numerical Solution
- Verification and Validation of Results

with these results a change in design may happen. Finally, with the theoretical, experimental, and numerical results the factors that induced if there is difference in these three results will be determined.

RESULTS AND DISCUSSION

Assumptions

First, the following assumptions were established:

- Steady flow
- Kinetic and potential energy changer negligible
- Specific heat of the fluid is constant
- Axial heat conduction along the tube negligible
- Outer surface of the heat exchanger is perfectly insulated
- Streamline
- No fouling was considered

Parameters

Hot Water Tube

- Type of fluid = Water
- Material of pipe = PVC
- Inlet temperature = 60°C
- Outlet temperature = 58.5°C
- Velocity inlet = 0.185m/s

Cold Water Tube

- Type of fluid = Water
- Material of pipe = PVC
- Inlet temperature = 10°C
- Outlet temperature = 12.8°C
- Velocity inlet = 0.098m/s

PVC Thermal Properties

- Density = 1330 kg/m³
- Specific Heat Capacity = 880 J/kg*K
- Thermal Conductivity = 0.2 W/m*K

Cooper Thermal Properties

- Density = 8960 kg/m³
- Specific Heat Capacity = 389 J/kg*K
- Thermal Conductivity = 385 W/m*K



Figure 3

Initial Model Development of Double Pipe Heat Exchanger

Double Pipe Heat Exchanger First Simulation Results

The present work involves numerical analysis of the heat exchanger with the design parameters established above. The first simulation performed with the double pipe heat exchanger with length at 2.748 meters the results are shown in Table 1 with the biggest difference in the Heat Transfer of 2.70%. In this case, even though the difference is minor, the results were verified and validated, and adjustments were done in the calculation of the convective heat transfer of the cold water with this change, a new and final design was developed.

Table 1
First Simulation Performed-Difference in Theoretical and Simulation values length of heat exchanger 2.748m

Theoretical	Values	Simulation	Values	Difference %
Heat Transfer in Watts	618.46	Heat Transfer in Watts	601.7887	2.70%
Outlet Temperature Cold Water in Kelvin	285.95	Outlet Temperature Cold Water in Kelvin	285.7473	0.07%
Outlet Temperature Hot Water in Kelvin	331.65	Outlet Temperature Hot Water In Kelvin	331.65	0%

Double Pipe Heat Exchanger Second Simulation Results

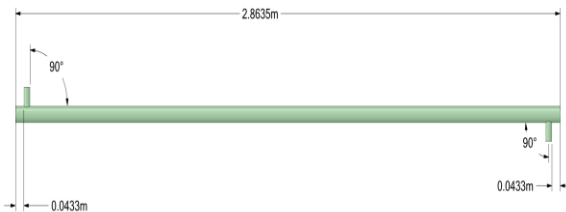


Figure 3
Double Pipe Heat Exchanger Geometry Final Design (Front View)

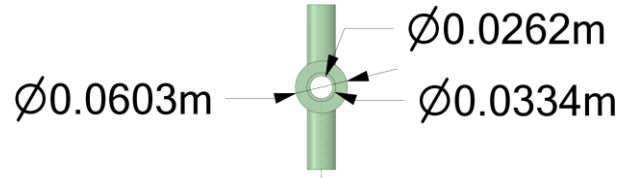


Figure 4
Double Pipe Heat Exchanger Geometry Final Design (Left View)

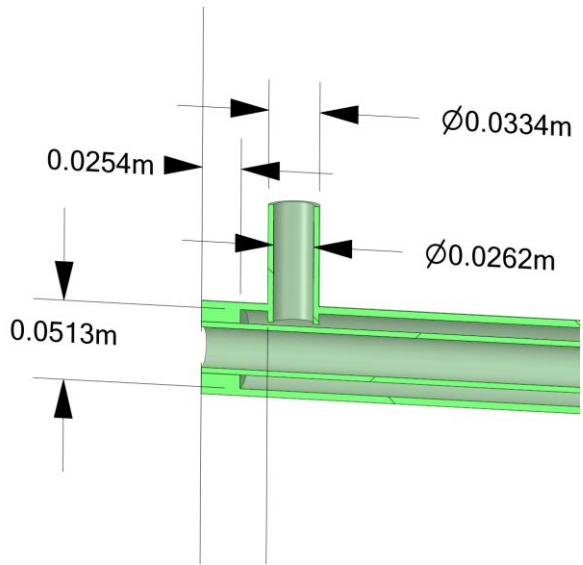


Figure 5
Double Pipe Heat Exchanger Geometry Final Design (Detailed Section)

After correcting the value of the convective heat transfer of the cold water, the length of the heat exchanger changes from 2.748m to 2.8635m and additional detailed of the model are shown in Figure 3, 4 and 5. With this geometry that was prepared in ANSYS the volume mesh as shown in Figure 6 was generated in ANSYS FLUENT with a mesh size of 257,434 and a minimum orthogonal of 0.15 which is acceptable utilizing the spectrum in Figure 7.

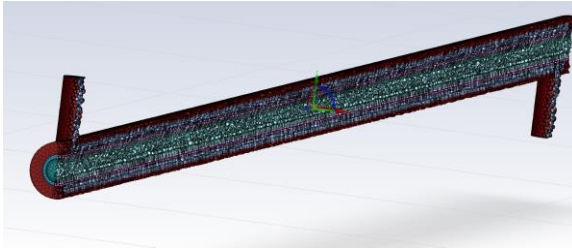


Figure 6

Volume Mesh Generated with a total of 257,434 cells



Figure 7

Mesh Orthogonal Quality Metrics Spectrum [4]

Finally, the initial conditions were coded in the numerical simulation and the results were generated with a difference in heat transfer of 0.22%, converging at 152 iterations additional information is shown in Table 2.

Table 2

Second Simulation Performed-Difference in Theoretical and Simulation values length of heat exchanger 2.864m

Theoretical	Values	Simulation	Values	Difference %
Heat Transfer in Watts	18.46	Heat Transfer in Watts	17.09	0.22%
Outlet Temperature Cold Water in Kelvin	85.95	Outlet Temperature Cold Water in Kelvin	85.30	0.23%
Outlet Temperature Hot Water in Kelvin	31.65	Outlet Temperature Hot Water in Kelvin	31.59	0.02%

In Figure 8, 9, 10, 11, 12, and 13 the temperature and pressure distribution in the inlets and outlets of the hot and cold water is shown. As the cold fluid start to flow inside the outer pipe, the temperature starts to increase until it comes out from the outer pipe. This can be seen in Figure 10 and 11.

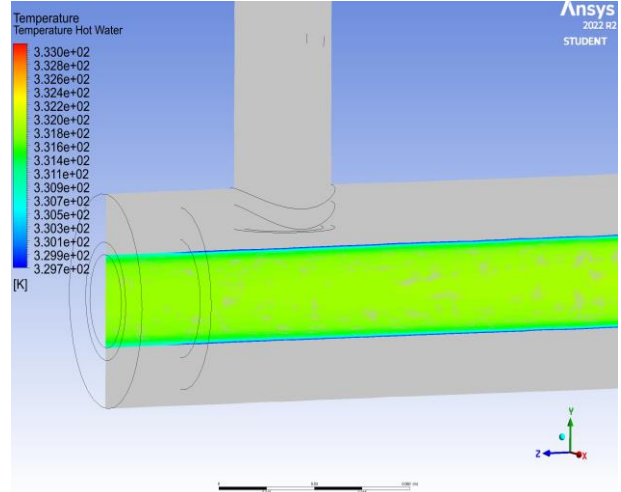


Figure 8

Outlet Temperature Variations of the Hot Water with inner tube material PVC

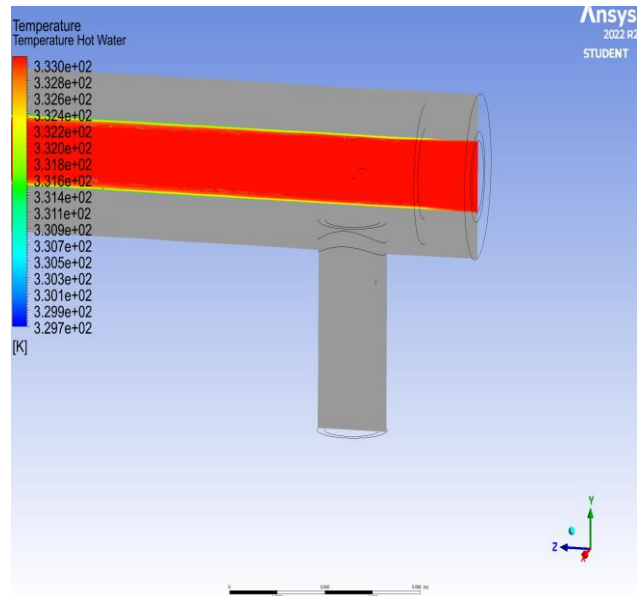


Figure 9

Inlet Temperature Variations of the Hot Water with Inner Tube Material PVC

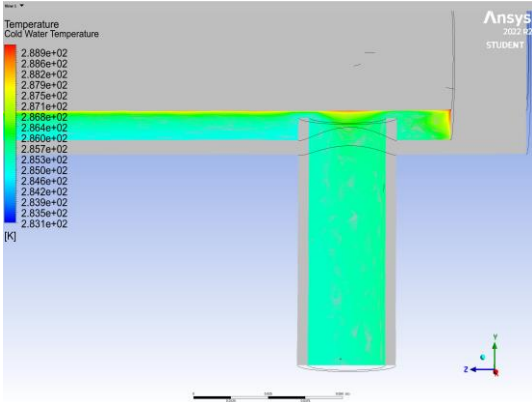


Figure 10
Outlet Temperature Variations of the Cold Water with Inner Tube Material PVC

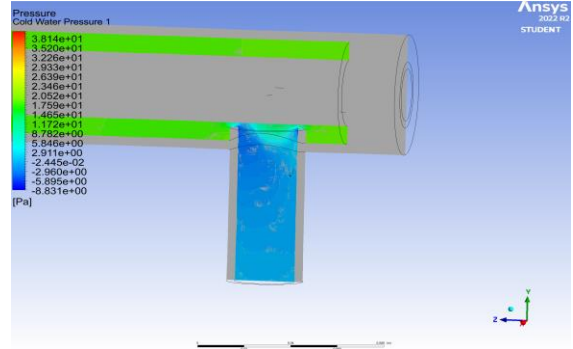


Figure 13
Outlet Pressure Variations in Pascals of the Cold Water with Inner Tube Material PVC

Double Pipe Heat Exchanger Third Simulation Results changing inner tube material to copper

After the verification and validation of the results of the second simulation, a third simulation was performed changing the material of the inner tube to copper, maintaining the same initial conditions; the results are shown in Table 3. The table indicate the difference in results when using a material like copper with better thermal conductivity. In Figure 14, 15, 16, 17, 18, 19, 20, and 21 the temperature and pressure distribution in the inlets and outlets of the hot and cold water is shown with the inner tube material copper. As the hot fluid start to flow inside the inner pipe, the temperature starts to decrease until it comes out from the inner pipe; this can be seen in Figure 16 and 17.

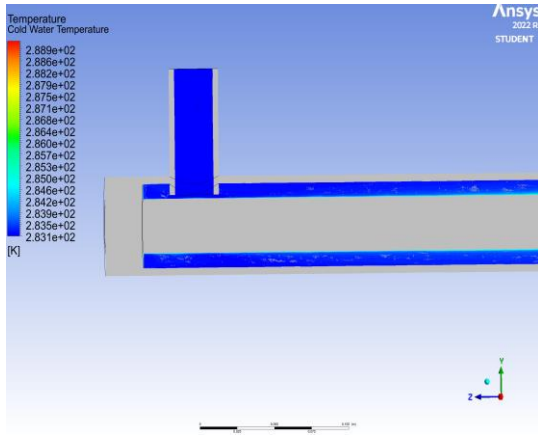


Figure 11
Inlet Temperature Variations of the Cold Water with Inner Tube Material PVC

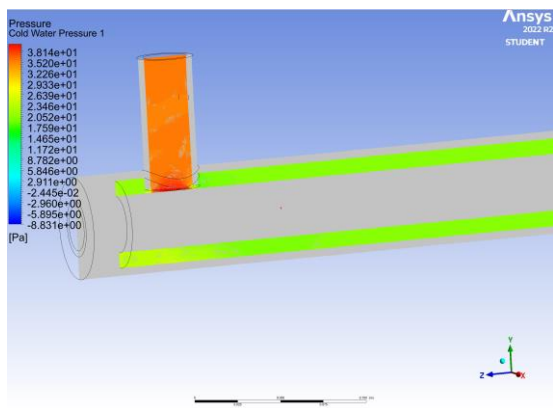


Figure 12
Inlet Pressure Variations in Pascals of the Cold Water with Inner Tube Material PVC

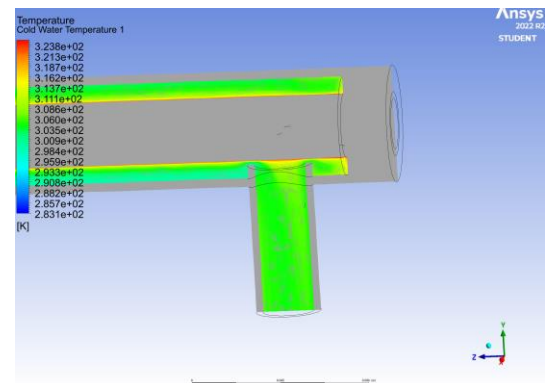


Figure 14
Outlet Temperature Variations of the Cold Water with Inner Tube Material Copper

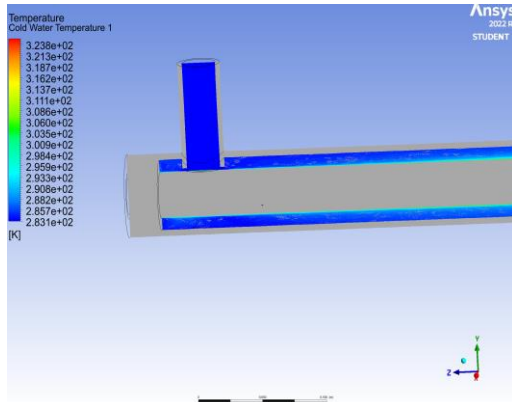


Figure 15

Inlet Temperature Variations of the Cold Water with Inner Tube Material Copper

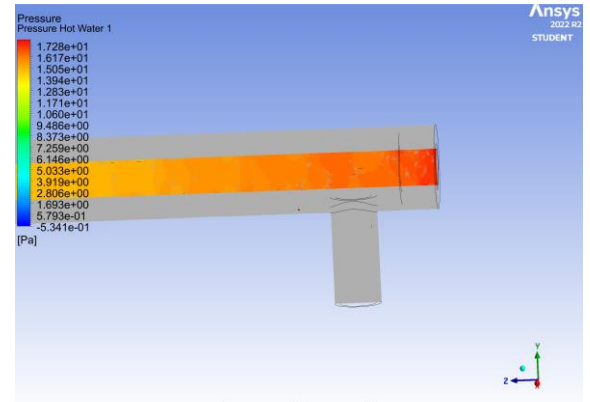


Figure 18

Inlet Pressure Variations of the Hot Water with Inner Tube Material Copper

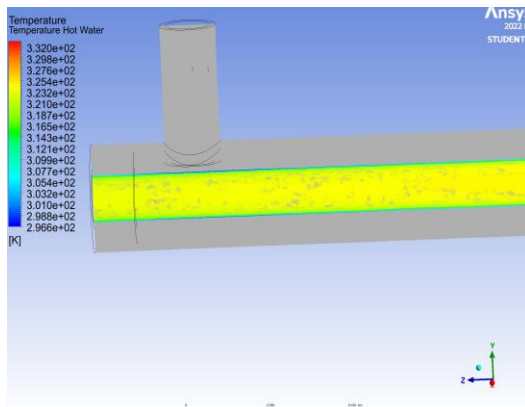


Figure 16

Outlet Temperature Variations of the Hot Water with Inner Tube Material Copper

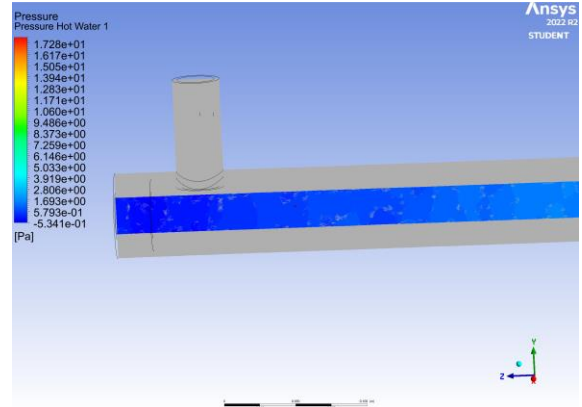


Figure 19

Outlet Pressure Variations of the Hot Water with Inner Tube Material Copper

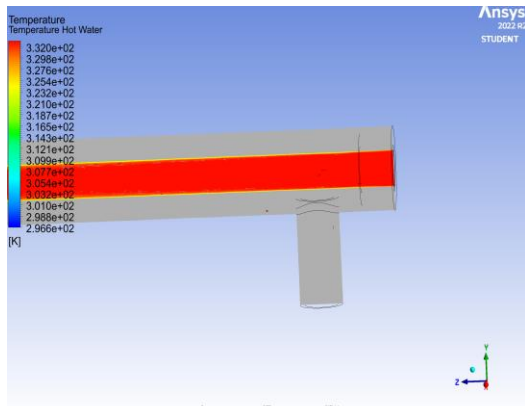


Figure 17

Inlet Temperature Variations of the Hot Water with Inner Tube Material Copper

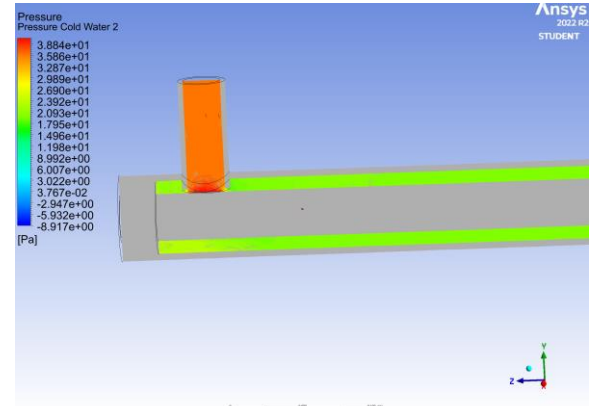


Figure 20

Inlet Pressure Variations of the Cold Water with Inner Tube Material Copper

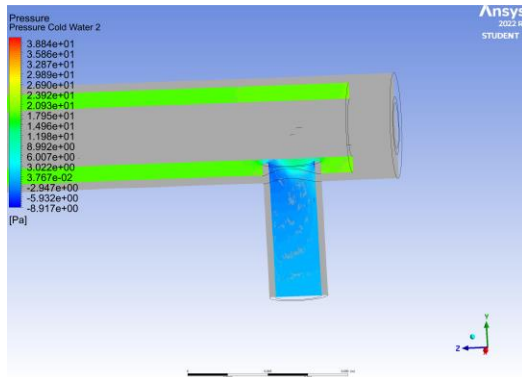


Figure 21
Outlet Pressure Variations of the Cold Water with Inner Tube Material Copper

Table 3
Comparing Simulation Results Changing the Inner Tube Material Length of Heat Exchanger 2.864m

Simulation with inner tube material PVC	Values	Simulation with inner tube material Copper	Values	Increase
Heat Transfer in Watts	617.09	Heat Transfer in Watts	4761.15	4,144.06
Outlet Temperature Cold Water in Kelvin	285.30	Outlet Temperature Cold Water in Kelvin	304.90	19.6
Outlet Temperature Hot Water in Kelvin	331.59	Outlet Temperature Hot Water in Kelvin	321.08	10.51

CONCLUSION

Despite the design of the double pipe heat exchanger is the simplest of all the heat exchangers, optimizing it can be difficult. In this case, based on the simulation analysis, can be concluded that:

- Numerical Simulation analysis can be a useful tool to find errors in the theoretical calculation.
- Utilizing numerical simulation, a 2% difference in the heat transfer value was achieved.
- After having the model verified and validated, optimizing is easier in the change of the inner tube material to copper; there is an increase in heat transfer of 4,144,06 W because copper is a

material with a high thermal conductivity, which makes perfect sense compared to PVC.

- The visualization numerical simulation gives better understanding of the behavior of the fluid.
- The linearization and discretization errors are big factors that can induced an unwanted difference in the results.

REFERENCES

- [1] B. Ramakrishan, "Thermal Analysis of Double Pipe Heat Exchanger with Different Mass Flowrates," *International Journal of Computational Engineering Research*, vol. 08, no. 2, pp. 50-55, 2018.
- [2] G. S. K. J. Mate Petrik, "Optimal Design of Double-Pipe Heat Exchanger," in *Advances in Structural and Multidisciplinary Optimization*, Springer International Publishing, 2018.
- [3] A. G. A. C. Rifah AMalia, "Numerical study on Heat transfer and Pressure Drop Characteristic in Double Pipe Heat Exchanger," in *International Electronics Symposium*, 2020.
- [4] S. T. C. N Fatcgurrohman, "Performance of hybrid nano-micro reinforced mg metal matrix composites brake calliper: simulation approach," in *4th International Conference on Mechanical Engineering Research*, Malaysia, 2017.