International Journal of Environmental Sciences
ISSN: 2229-7359
Vol. 11 No. 18s, 2025
https://theaspd.com/index.php

Numerically Analyzing A Heat Transmission In Blood Using A Cardioplegia Concentric Tube Heat Exchanger During Open Cardiac Surgery

Anbuchezian Ashokan[1], Senthil Murugesan[2], Ravikumar Muthaiyan[3], Sivashankar Arumugam[4], Perumal Saravanan[5*], Senthilkumar Ramuvel[6], Mohankumar Pachamuthu[7], Silambarasan Rajendran[2,8],
[1]Department of Civil Engineering, Annapoorana Engineering College (Autonomous), Salem, 636308, Tamil Nadu, India.
[2]Department of Mechanical Engineering, Annapoorana Engineering College (Autonomous), Salem, 636308, Tamil Nadu, India.
[3]Department of Mechanical Engineering, Bannari Amman Institute of Technology (Autonomous), Sathyamangalam, Erode, 638401, Tamil Nadu, India.
[4]Department of Mechanical Engineering, Dhanalakshmi Srinivasan College of Engineering (Autonomous), Coimbatore, 641105, Tamil Nadu, India.
[5]Department of Mechanical Engineering, Muthayammal Engineering College (Autonomous), Rasipuram, 637408, Tamil Nadu, India.
[6]Department of Mechtronics Engineering, Muthayammal Engineering College (Autonomous), Rasipuram, 637408, Tamil Nadu, India.
[7]Department of Mechanical Engineering, Jayalakshmi Institute of Technology, Thoppur, 636352, Tamil Nadu, India.
[8]Department of Mechanical Engineering, Saveetha School of Engineering, Saveetha Institute of Medical and Technical Sciences, Chennai, Tamil Nadu, India.
[*]**Corresponding Author Email: autperumal@gmail.com**

ABSTRACT:
A blood heat exchanger primary objective is to reduce the amount of time a patient's blood needs to cool before open heart surgery. Blood behaves non-Newtonian. Hence, some assumptions have to be made in order to finish the computations. The smoothest surface possible is required for the material in touch. Because blood is brittle, an adequate design must have a low pressure drop and a high heat transfer rate to accommodate the needed quick temperature change. The thermal, dynamic, and pressure drop analyses of a blood heat exchanger will be performed in this work. The goal of blood cooling is to swiftly decrease body temperature, which in turn leads to a reduction in metabolic demands. The heat exchanger water component is set to run at a fixed temperature of 15°C and 0.2 Kg/s flow rate. Blood has a temperature of 37°C and a flow rate that ranges from 0.03 Kg/s. The performance of the blood heat exchanger and the systemic flow rate are the factors that determine the rate of hypothermic induction, regardless of other factors. The numerical analysis's findings can be used in efforts to cause and regulate hypothermia in order to induce it during open cardiac surgery.

1. INTRODUCTION:

One cannot sustain a circulatory stoppage for longer than three to four minutes at normal body temperature without suffering catastrophic or lethal brain damage. The origins of severe hypothermia can be traced back to the evolution of open cardiac surgery. Open cardiac surgery was uncommon prior to the 1939–1945 war. After the war, hypothermia a method of bringing the body temperature down to around 30°C saw significant success. Surface cooling was the technique used, typically with the patient submerged in an ice bath or with fans blowing cold air over their body. A heat exchanger is a device that lowers the patient's blood temperature during open heart surgery and raises it when the procedure is finished. Appropriate anesthesia is required to overcome the patient's natural shivering response, which is a result of muscles and skin reacting to try and return to normal. During open heart surgery, the blood heat exchanger is utilized. The blood heat exchanger was created to reduce the amount of time that is often needed to rewarm a patient after open heart surgery and to cool them down before the procedure. An ice pack or a refrigerated blanket will bring a patient's body temperature down. Which needed to be under anesthesia for one to two hours before the procedure could start. In just a few minutes, the body temperature can be safely decreased thanks to the blood heat exchanger. Similarly, a patient can be rewarming to normal body temperature in 10 to 15 minutes instead of 3 to 4 hours by using this heat exchanger. Blood loss for the patient and disruption to the procedure are the reasons why open-heart surgery requires stopping the flow of blood through the heart. Complete blood removal from the heart and transfer of the blood into the heart-lung machine allows for this to happen. It is comprised of a water circuit that is isolated from the blood circuit by the heat exchanger surface. The temperature of the water out is represented by TWO, the temperature of the water in by TWI, and the temperature of the blood in by TBI and out by TBO. Ahuja, A. S., & Hendee, W. R (1), developed a thermal design of a counter current heat exchanger for heating or cooling blood is presented, together with experimental data and their agreement with theoretical predictions. Consolo, F, et.al.,(2) created a numerical model to aid in the design of a plastic hollow-fiber bundle blood heat exchanger (BHE) integrated within the INSPIRE™ blood oxygenator (OXY) for cardiac by-pass surgeries by using CFD. Older red blood cells are therefore not harmed by this blood warmer, which warms blood effectively at high flow rates not even when they are exposed to the 40 degrees Celsius heat exchanger for an extended period of time evaluated by Kruskall, M. S.et.al, (3). A heat exchanger has been shown to induce profound hypothermia in both clinical and experimental settings by BjöUrk,V.O (4). Robert R Harrison and Jr Leonard F Waldman (5), created a new type of heat exchanger for blood and it has been filed a patent in US Patent office. A new type of heat exchanger has been developed for Open Heart Surgery and Extracorporeal Circulation by Emmons, W.O and Sacca, D.B (6) and Brown,I.W.,et.al. (7). The functionality of a water-jacketed heat exchangers the Buckberg Shiley, Shiley BCD, and Bentley HE-100 have been compared to a polyethylene coil submerged in ice. By comparing the blood outflow temperatures at the same inflow temperature under a range of test settings that mimicked those found in clinical practice, performance was assessed and the ice coil system offers the benefits of economy, efficiency and simplicity by Newman, M.et.al (8). Drummond, M (9), studied about various myocardial heat exchangers are employed in cardioplegic systems during heart operations such as Aluminum, polypropylene hollow fiber, bellows type, and stainless steel tubes. Raman, M.,et.al, (10), numerically and experimentally studied about, the waste heat recovery from diesel engine exhaust gas through triple concentric tube heat exchanger with various tubes for various renolds numbers.Perumal saravanan and Mohan raman (11) numerically and experimentally studied about, the waste heat recovery from diesel engine exhaust gas through triple concentric tube heat exchanger with plain tube for various engine load conditions. Senthilkumar et al. (12) has been numerically studied about, the spherical shape dimpled tube through the Triple concentric tube heat exchanger and compared with plain tube via CFD Simulation. Perumal, S et al. (13) Studied about different heat transfer enhancement techniques by using different types of nanofluids such as AL2O3, CUO2...etc and studied about various thermal behaviours like heat transfer rate, pressure drop, friction factor, pressure drop, effectiveness and heat transfer coefficient. K.Senthilkumar et al. (14) has been numerically studied on dimpled concentric double tube heat exchanger with AL2O3 nanofluid and compared with smooth tube. In this research, numerically analyzing a heat transmission in blood using a cardioplegia concentric tube heat exchanger while open cardiac surgery. By using ANSYS Fluent the heat transfer analysis was made. In cardioplegia heat exchanger outside tube the water was flowing with 15°C and inside tube the blood was flowing with 37°C. The heat exchanger was made by stainless steel 305.The dimensions of the heat exchanger is 1.5 inches outer tube outer diameter, 0.20 inches inner tube inner diameter and 15.75 inches length and thickness of the each tube is 0.02 inches, The above measurements has been taken from Robert R Harrison and Jr Leonard F Waldman (5).

2. MATERIAL AND METHODS

Stainless steel tubes or bellows enclosed in a plastic shell are commonly used to construct cardioplegia concentric tube heat exchanger. To heat or chill blood or cardioplegia solution, they make advantage of water flowing within the shell. Further details regarding concentric tube heat exchangers and CCTHE are provided below: Often called double-pipe heat exchangers, these units consist of a larger outer tube encircling a smaller inner tube. A narrow, conductive metal tube separates the two fluids, allowing for quick heat exchange. For exchanger tubes, the most conductive materials are copper and copper/nickel. Although they function similarly, higher alloys, carbon steel, and stainless steel are marginally less efficient. According to in vitro experiments, shell and tube heat exchangers composed of polypropylene, aluminum, and stainless steel tubes. The thermal conductivity of aluminium is 200W/m.K and the thermal conductivity of polymer is 0.2 W/m.k. Here are the CCTHE specifications: Blood volume is 175 cc, the outside tube diameter is 54 mm, the heat exchanger is 470 mm long, and it weighs 2.7 kg.

2.1. Geometric Model of the Cardioplgia Heat Exchanger

A concentric tube heat exchanger was designed to mimic the structure commonly used in open cardiac surgery. The dimensions and geometry were obtained from commercially available heat exchangers, with specifications adapted for numerical simulation. The model consisted of an inner tube carrying cardioplegia solution and an outer tube circulating a heating medium (e.g., water or saline). The Modelling of cardioplegia concentric tube heat exchanger is shown in figure 1.

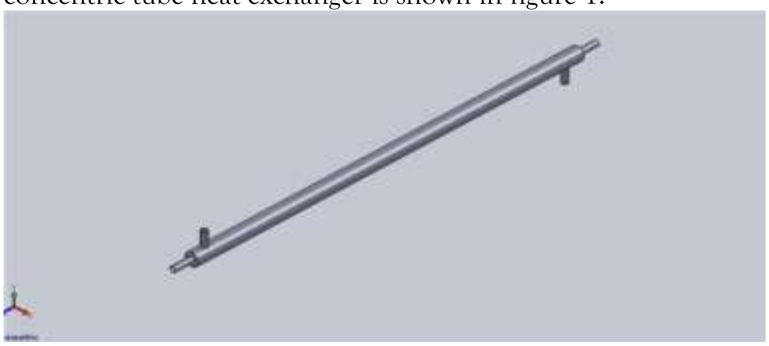

Figure 1. Modelling of Cardioplegia Concentric Tube Heat Exchanger (CCTHE)

2.2 Meshing

A structured mesh was generated for the concentric heat exchanger geometry, ensuring finer grids near the walls to resolve boundary layer effects. A mesh independence study was conducted to ensure accuracy without excessive computational cost.The Meshing of cardioplegia concentric tube heat exchanger is shown in figure 2.

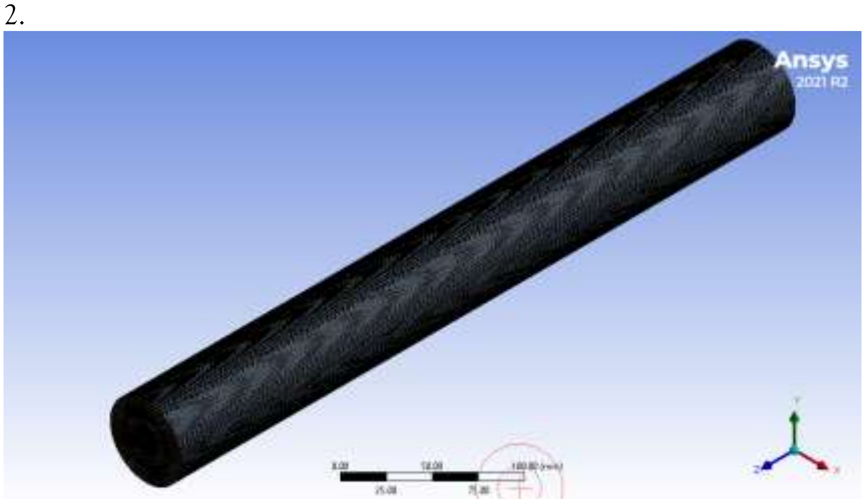

Figure 2.Meshing of cardioplegia concentric tube heat exchanger

2.3. Boundary Conditions and Assumptions

Blood: The physical properties (density, specific heat, and thermal conductivity) were modelled based on standard values from literature, accounting for its non-Newtonian behavior. Cardioplegia solution: Properties similar to saline solution were considered, with standard values for thermal conductivity, specific heat, and density. Heating medium: Water properties were used for the external circulating fluid. The physical Properties of blood at 32°C and Water at 15°C is shown in following table 1.

Table 1. Physical Properties of blood at 32°C and Water at 15°C

Physical Properties	**Blood**	**Water**
Density (ρ)	1055 kg/m^3	999.19 kg/m3
Dynamic Viscosity (μ)	0.00045 kg/m s	0.001170 kg/m s
Thermal Conductivity (k)	0.426 W/m.k	0.6 W/mK
Specific Heat Capacity (cp)	3330 J/kg.k	4187 J/kg.k
Prandle Number (Pr)	3.52	6.9

ANSYS Fluent was employed for computational fluid dynamics simulations to analyze heat transfer. Pre-processing tools included ANSYS Design Modeler for geometry creation and ANSYS Meshing for grid generation. Inlet temperatures for the blood, cardioplegia solution, and heating medium were defined based on clinical scenarios. Steady-state heat transfer was assumed for simplicity. Blood flow was modelled as laminar due to the low Reynolds number in such medical devices. Walls were assumed to have no-slip conditions. Inlet mass flow rates were specified based on typical flow rates observed during cardioplegia administration. The heating medium's inlet temperature was set to the desired value for maintaining blood temperature. The outlet conditions were defined as temperature and pressure outlets. Input parameters of blood and water are shown in following table 2.

Table 2. Input Parameters of Blood and Water

Parameters	**Hot Fluid (Blood)**	**Cold Fluid (Water)**
Inlet temperature(°C)	37	15
Outlet temperature(°C)	35.17	16.33
Mass flow rate (Kg/s)	0.03 to 0.0833	0.2

2.4 Solver Setup:

The pressure based solver is suitable for in-compressible flow, such as the blood and water mediums and steady-state heat transfer and fluid flow. The absolute velocity formulation is selected because it handles the velocity fields based on absolute pressure and velocity values, which is more appropriate for heat exchangers with steady-state laminar flow. Use appropriate discretization methods for temperature and velocity fields. For velocity, use the second-order upwind scheme for better accuracy. For temperature, use second-order upwind for the temperature equation to capture the temperature gradients accurately in the fluid. SIMPLE (Semi-Implicit Method for Pressure-Linked Equations) algorithm is used pressure-velocity coupling method for steady-state simulations. It provides a balance between computational efficiency and accuracy, especially for in-compressible flow in a heat exchanger. After running the simulation, extract temperature profiles along the radial direction of both the blood) and the water. Plot the temperature distribution along the length of both the inner and outer tubes to identify areas of high or low heat transfer. The temperature gradient between the inner (blood) and outer (water) fluids is key. A steep temperature gradient indicates efficient heat transfer, while a shallow gradient suggests poor heat exchange. The ideal heat exchanger should have a significant temperature drop in the water and a temperature rise in the blood.

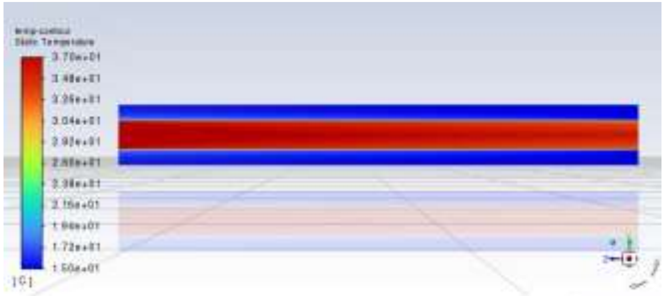

Figure 3.Temperature Distribution of cardioplegia concentric tube heat exchanger

The Figure 3, illustrates a static temperature contour plot derived from a Computational Fluid Dynamics (CFD) simulation, likely depicting the temperature distribution within a rectangular duct or channel. The color scale varies from 15.0°C (blue) to 37.0°C (red), with the central area appearing in red, signifying elevated temperatures, while the peripheral regions adjacent to the walls exhibit cooler temperatures in blue. This indicates that the core flow retains a greater amount of heat, whereas the temperature decreases near the boundaries, potentially due to heat loss or cooler wall conditions. The smooth and symmetrical temperature gradient suggests a probable laminar flow regime. Such a plot is commonly utilized in the thermal analysis of applications such as heat exchangers, electronic cooling systems, or duct flow systems. It aids in visualizing how heat is distributed or transferred throughout the domain and can be instrumental in assessing thermal performance and optimizing design.

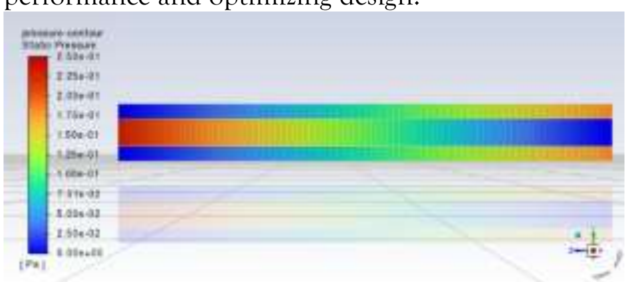

Figure 4.Pressure Distribution of cardioplegia concentric tube heat exchanger

The Figure 4, illustrates a static pressure contour plot derived from a Computational Fluid Dynamics (CFD) simulation, which demonstrates the pressure distribution within a rectangular duct or channel. The color gradient spans from 0.00 Pa (blue) to 0.25 Pa (red), indicating a gradual decrease in pressure from the inlet to the outlet. The high-pressure area near the inlet is represented in red, transitioning smoothly through orange, yellow, and green towards the outlet, depicted in blue. This linear pressure gradient is characteristic of steady, in-compressible internal flow—most likely within a laminar regime—signifying a consistent loss of momentum due to wall friction. This visualization is instrumental in analyzing flow resistance and assessing pressure drop across the domain, which is essential for the design of efficient flow systems such as ventilation ducts, micro channels, or fluid transport pipelines.

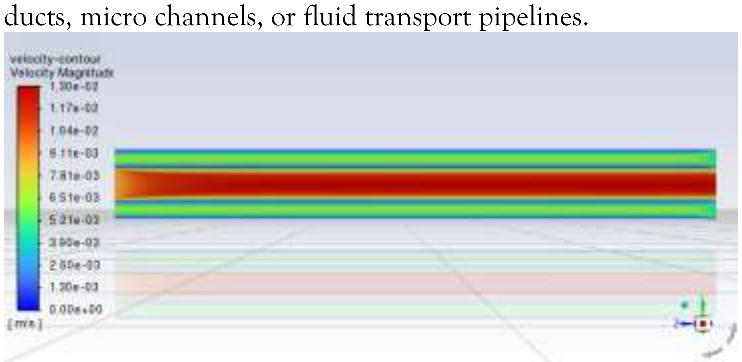

Figure 5. Velocity Distribution of cardioplegia concentric tube heat exchanger

The Figure 5, presents a contour plot of velocity magnitude derived from a Computational Fluid Dynamics (CFD) simulation, demonstrating the variation of fluid velocity within a rectangular duct or channel. The velocity scale ranges from 0.00 m/s (indicated in blue) to 0.013 m/s (indicated in red), with the peak velocities occurring in the central core region of the flow, depicted in red, and progressively diminishing towards the walls, where the velocity nears zero due to the no-slip boundary condition. This parabolic velocity profile is typical of fully developed laminar flow through a duct, where viscous effects prevail and generate velocity gradients from the center line to the walls. The image effectively illustrates the distribution of fluid flow across the channel and can be utilized to analyze flow behavior, assess wall shear effects, and optimize channel or duct designs for reduced flow resistance or targeted flow rates in applications such as microfluidic devices, heat exchangers, or biomedical flow systems.

3. RESULTS AND DISCUSSION

The provided information relates to a heat exchange operation where blood serves as the hot fluid and water acts as the cold fluid. The heat transfer rate (Q_h) from the blood is determined using the equation with the mass flow rate $\dot{m}_h$ being 0.03 kg/s, the specific heat capacity of blood at 3330 J/kg·K, and the temperature difference calculated as (37 - 35.17)°C. Plugging in these figures results in a heat transfer rate of 182.77 W. The heat capacity rate of the hot fluid (C_h), which reflects the fluid's capacity to absorb or release heat, is calculated to be 99.9 W/K. Meanwhile, the heat capacity rate of the cold fluid (C_c), derived from a mass flow rate of 0.2 kg/s and the specific heat capacity of water at 4182 J/kg·K, comes out to 836.4 W/K. From these calculations, the minimum heat capacity rate (C_{min}) is identified as 99.9 W/K, and the maximum (C_{max}) is 836.4 W/K. This differentiation is crucial for evaluating the performance of the heat exchanger. To assess the performance from the perspective of the cold fluid, the heat transfer rate (Q_c) is calculated based on the increase in outlet temperature of the cold water. With the cold water temperature rising from 15°C to 16.33°C, the computed Q_c is 222.05 W, which is marginally greater than Q_h, potentially due to variations in measurement or unaccounted heat losses/gains. These calculations together offer a fundamental insight into the thermal dynamics and efficiency of the heat exchanger system.

Heat Transfer Rate (Q_h) – Blood:

$$Q_h = \dot{m}_h \cdot Cp \cdot \Delta T = 0.03 \cdot 3330 \cdot (37 - 35.17) = 182.77 \text{ W} \quad (1)$$

Heat Capacity Rate (C_h) – Blood:

$$C_h = \dot{m}_h \cdot Cp = 0.03 \cdot 3330 = 99.9 \text{ W/K} \quad (2)$$

Heat Capacity Rate (C_c) – Water:

$$C_c = \dot{m}_c \cdot Cp = 0.2 \cdot 4182 = 836.4 \text{ W/K}$$

Minimum and Maximum Heat Capacity Rates:

$$Cmin = \min(C_h, C_c) = 99.9 \text{ W/K} \quad (3)$$

$$Cmax = \max(C_h, C_c) = 836.4 \text{ W/K}$$

Outlet Temperature of Cold Fluid (Verification):

$$Qc = \dot{m}_c . Cp . \Delta T = 0.2 \cdot 4182 \cdot (16.33 - 15) = 222.05 \text{ W} \quad (4)$$

Table 3. Heat Transfer Rate and Pressure Drop for Different Mass Flow Rates

Mass Flow Rate (kg/s)	Heat Transfer Rate (W)	Pressure Drop (Pa)	Effectiveness (ε)
0.030	182.77	0.16	0.18295
0.045	296.4	0.36	0.19779
0.060	395.3	0.64	0.19784
0.075	494.1	1.00	0.19783
0.0833	548.5	1.24	0.19759

3.1. Heat Transfer Rate vs Mass Flow Rate

The chart depicts the connection between mass flow rate and heat transfer rate within a thermal system, clearly indicating a trend where heat transfer increases with greater mass flow rates. At a mass flow rate of 0.030 kg/s, the system realizes a heat transfer rate of approximately 182.77 W. As the mass flow rate escalates to 0.045 kg/s, the heat transfer rate significantly climbs to around 296.4 W. When the mass flow rate further rises to 0.060 kg/s, the heat transfer rate achieves 395.3 W, and at 0.075 kg/s, it increases to 494.1 W. Ultimately, at the peak recorded mass flow rate of 0.0833 kg/s, the heat transfer rate reaches its maximum at 548.5 W. This consistent and nearly linear growth affirms that a higher flow of fluid facilitates more energy transfer, consistent with the heat transfer equation. The data highlights a direct correlation between mass flow rate and heat transfer rate, reinforcing the idea that boosting fluid flow through a heat exchanger improves its thermal efficiency.

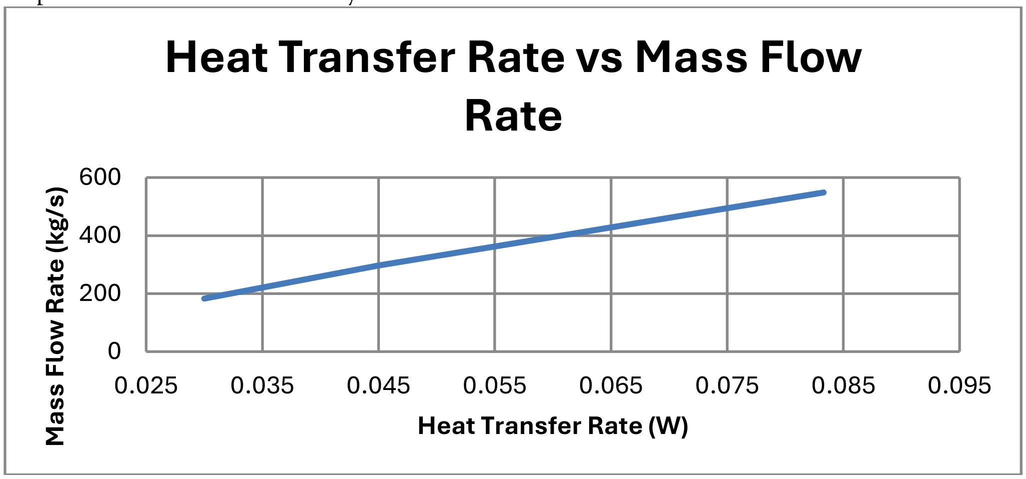

Figure 6. Heat Transfer Rate vs Mass Flow Rate

3.2. Pressure Drop vs Mass Flow Rate

The chart labeled Pressure Drop vs Mass Flow Rate demonstrates the connection between the pressure drop (measured in Pascals) and the mass flow rate (in kg/s) within a fluid system. It is evident that there is a non-linear, increasing trend which shows that the pressure drop rises more swiftly as the mass flow rate increases. This pattern is common in fluid dynamics, especially in internal flows through pipes or channels, where frictional losses and turbulent effects become more pronounced at elevated flow rates. At first, with lower flow rates, the pressure drop increases steadily; however, as the flow rate escalates, the pressure drop surges more sharply, indicating a rise in resistance due to turbulence. The graph implies that sustaining higher flow rates necessitates considerably more energy to counteract the pressure losses within the system. Understanding this relationship is vital for the design and enhancement of fluid transport systems, as excessive pressure drops can lead to decreased efficiency and heightened operational expenses.

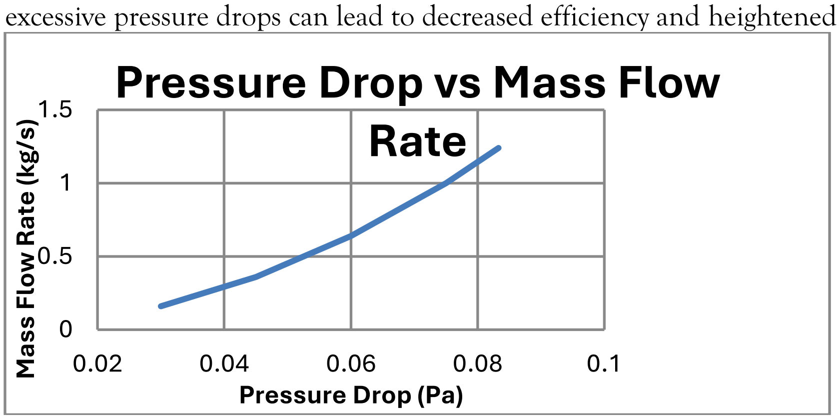

Figure 7. Pressure Drop vs Mass Flow Rate

3.3. Effectiveness vs Mass Flow Rate

According to the chart, the efficacy first rises dramatically with mass flow rate, reaching roughly 19.9% at 0.045 kg/s after starting at 18.4% at 0.030 kg/s. After this, the efficiency is almost constant with a tiny negative tendency towards the end as the mass flow rate goes further (up to 0.0833 kg/s).

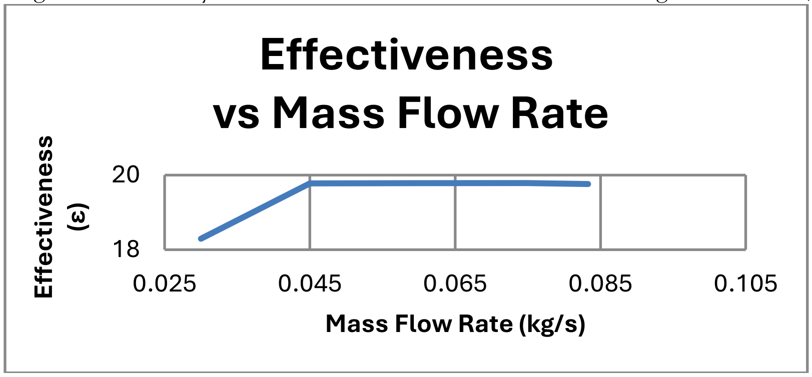

Figure 7. Effectiveness vs Mass Flow Rate

Heat exchange performance is greatly enhanced by increased flow at lower mass flow rates, presumably as a result of improved thermal contact and more effective energy transfer.After a particular flow rate (around 0.045 kg/s), the system reaches its maximum efficiency, and further mass flow rate increases barely improve performance. This plateau denotes a saturation point, beyond which the heat exchanger efficiency cannot be appreciably increased.

4. CONCLUSION

This study evaluated heat transfer rate, pressure drop, and effectiveness to investigate how mass flow rate affects a heat exchanger thermal performance. The findings clearly show that the mass flow rate and the heat transfer rate are positively correlated. In particular, the associated heat transfer rate increased consistently from 182.77 W to 548.5 W as the mass flow rate increased from 0.030 kg/s to 0.0833 kg/s, suggesting improved thermal energy transmission with increasing fluid flow.However, this gain in heat transport comes at the cost of higher hydraulic resistance. The system-wide pressure loss increased nonlinearly, from 0.16 Pa at 0.030 kg/s to 1.24 Pa at 0.0833 kg/s, due to increased friction and turbulence at high flow rates. The system showed a discernible improvement in heat exchanger performance up to a mass flow rate of 0.045 kg/s, where it peaked at roughly 19.9%, up from 18.4% at 0.030 kg/s. A performance saturation point was indicated by the marginal changes that followed incremental increases in mass flow, with efficacy gradually decreasing towards the highest evaluated rate.All things considered, the results show that although raising the mass flow rate greatly improves heat transfer, it also causes greater pressure reductions and only modest increases in efficacy over a certain point. These understandings are essential for heat exchanger design and operation, because maximum system performance requires striking a compromise between fluid dynamic losses and thermal efficiency.

REFERENCES

1. Ahuja, A. S., & Hendee, W. R. (1978). Thermal design of a heat exchanger for heating or cooling blood. Physics in Medicine & Biology, 23(5), 937.
2. Consolo, F., Fiore, G. B., Pelosi, A., Reggiani, S., & Redaelli, A. (2015). A numerical performance assessment of a commercial cardiopulmonary by-pass blood heat exchanger. Medical Engineering & Physics, 37(6), 584-592.
3. Kruskall, M.S., Pacini, D.G., Malynn, E.R., & Button, L. N. (1990). Evaluation of a blood warmer that utilizes a 40 degrees C heat exchanger. Transfusion, 30(1), 7-10.
4. BjöUrk, V. O. (1960). An effective blood heat exchanger for deep hypothermia in association with extracorporeal circulation but excluding the oxygenator. The Journal of Thoracic and Cardiovascular Surgery, 40(2), 237-252.
5. Robert R Harrison and Jr Leonard F Waldman (1979), Heat Exchanger for Blood (U.S. Patent No.4, 177,816) https://patents.google.com/patent/US4177816A/en.

6. Emmons, W.O., and Sacca, D.B., "The Design and Development of a Blood Heat Exchanger for Open Heart Surgery", General Motors Engineering Journal, July - September 1958, p. 39.

7. Brown, I.W., Smith, W.W., and Emmons, W.O., "An Efficient Blood Heat Exchanger for Use with Extracorporeal Circulation", Surgery, Volume 44, No. 2, 1958, pp. 372-375.

8. Newman, M., Munsch, C., Mcmillan, J., Slavotinek, J., & Rosenfeldt, F. (1989). A comparison of heat exchangers for blood cardioplegia. Australian and New Zealand Journal of Surgery, 59(4), 343-346.

9. Drummond, M., Novello, W. P., Arruda, A. C. F. D., & Braile, D. M. (2003). Heat exchangers for cardioplegia systems: in vitro study of four different concepts. Artificial organs, 27(5), 465-468.

10. Raman, M., Saravanan, P., Muthusamy, S., & Subramaniam, S. (2022). Studies on diesel engine exhaust gas for retrieving the waste heat through Triple Tube Heat Exchanger (TTHE) through different tubes. Energy Sources, Part A: Recovery, Utilization, and Environmental Effects, 44(2), 4149-4164.

11. Perumal saravanan and Mohan raman., 2020 Experimental And Numerical Analysis Of Diesel Engine Exhaust Heat Recovery Using Triple Tube Heat Exchanger, Thermal Science, 2020:24; 525-531.

12. Kumar, K.S., Perumal, S., Mohan, R. and Kalidoss, K., 2016. Numerical Analysis of Triple Concentric tube Heat Exchanger using Dimpled Tube Geometry. Asian Journal of Research in Social Sciences and Humanities, 6(8), pp.2078-2088.

13. Perumal, S., Mohan, R., Sasidharan, S. and Venkatesh, K., 2017. Study On Concentric Tube Heat Exchanger With Different Nano Fluids For Enhancing The Heat Transfer: A Review., Imperial Journal of Interdisciplinary Research, Volume 3, Issue 9, ISSN 2454-1362. 682-688.

14. K.Senthilkumar,S.Perumal, P.Palanisamy., 2014 Numerical study on a concentric tube heat exchanger using dimpled tubes with al2o3 nanofluid., Australian journal basic and applied sciences, 8 (7), 185-193.

15.