DESIGN, PERFORMANCE EVALUATION AND VALIDATION OF REUSABLE BLOOD HEAT EXCHANGER FOR OPEN HEART SURGERY USING ANALYTICAL AND NUMERICAL ANALYSIS

Shashikumar M[1], Dr. M Shivashankar[2]
1(M Tech Student, Department of mechanical Engineering
Siddaganga Institute Of Technology Tumakuru -572103)
2 (Associate Professor Department of Mechanical Engineering
Siddaganga Institute Of Technology Tumakuru -572103)

Abstract:

Analytical data and their correlation with calculations from theory are presented for the thermal design of a counter flow heat exchanger for cooling blood. The thermal design considers the influence on the Reynolds number of blood and the heat exchanger variables such as the blood mass flow rate, tube diameter and length, and the thermal properties of blood. The data presented are compared with data from the literature and with forecasts from theory. Insofar as the design of a blood heat exchanger is concerned, flowing blood can be considered a single-phase fluid. Results of the numerical thermal design analysis are applicable to attempts to produce and control hypothermia for applications to induce hypothermia during open-heart surgery.

Keywords — Shell and Tube heat exchanger, blood and water.

1. INTRODUCTION

The blood heat exchanger is used in open heart surgery. The blood heat exchanger is developed to shorten the time normally required to cool a patient prior to open heart surgery and to rewarm the patient following surgery. The body temperature of a patient will be lower by either a refrigerated blanket or an ice pack. Which required from one to two hours under anaesthesia before the operation could begin. The blood heat exchanger allows the body temperature to be lowered safely in a very few minutes. And carefully controlled during the actual surgery. In a like manner, the heat exchanger allows rewarming of a patient to normal body temperature will lower within 10-15 minutes, as compared to 3-4 hours. In open-heart surgery, the blood flowing through the heart need to be stopped, as it will cause blood loss for the patient and disturbance during the operation. This is achieved completely removing the blood from the heart and passing it into the heart-lung machine.

The blood is now pumped through the heart-lung machine to every part of the patient's body. Since the blood is stopped from the heart, the oxygen exchange in the blood through the lungs also fails. In this case, the oxygen is also supplied from the heart-lung machine. Since the tubes of the heart-lung machine and the human veins differ in properties, the blood molecules will get damaged. In order to minimize the damage caused, the oxygen level in the blood has to be brought down to an optimum level. This is achieved by lowering the temperature of the blood, which in turn reduces the requirement of oxygen when it enters the heart-lung machine. By this way, we can ensure that the damage caused to the molecules of the blood is minimized up to a great extent. Hence, a device is necessary to lower the blood temperature. This is known as a blood heat exchanger. Traditionally, heat exchangers, development is carried out using the analytical method and experiments, which is time-consuming and expensive. Of late as powerful computational platforms are being realized on a continuous basis, the feasibility of using CFD analyses is undoubted for heat exchanger design.

2. LITERATURE SURVEY

Harrison has introduced blood heat exchangers for open heart surgery has become standard practice today. They are now employed on all heart-lung machines in the world. Either they are disposable or they are built into the disposable blood oxygenators. In addition, their use has been expanded to deep hypothermia for certain special surgery. Swan described Hypothermia is an extremely safe technic for open heart surgery in those conditions in which the reparative procedure can be accomplished in 10 minutes or less. Ahuja et al was presented for the thermal design of a counter-current heat exchanger for heating or cooling blood. The thermal design considers the influence on the Nusselt number of blood and the heat exchanger effectiveness of variables such as the blood flow rate, tube diameter, length, and the thermal properties of blood. Russell investigated the effective warming and infusion of blood is governed by factors which can be theoretically analyzed. How effectively blood can be heated depends on the dimensions of the tube and its ability to transfer heat. The output temperature also depends on the specific heat and thermal conductance of the liquid. However, the temperature of the infusion into the patient will be decreased by heat loss in the output line. The maximum flow which can be achieved is determined by the liquid's viscosity and the resistance of the tubing and veins. Edward et al made an attempt to design a shell and tube heat exchanger and calculated the shell side heat transfer coefficient, nusselt number, required heat transfer area, pressure drop at shell side and pressure drop at the tube side.

3. PROBLEM STATEMENT AND PROJECT OBJECTIVES

3.1 Problem statement

Most heat exchangers in open heart surgery have one or more of the following disadvantages.

i. They are bulky in size and having a high priming volume.

ii. Although some are ‘disposable’, most are difficult to clean and sterilize as well as to debubble when priming.

iii. Welded and other joints may be imperfect which leads to leaking.

iv. Manufacture is relatively time-consuming and costly.

V. These heat exchangers have more pressure drop and low heat transfer rate.

Vi. These heat exchangers are not reusable.

3.2 Project objectives

The objectives of this project are:

i. To make a feasibility study on types of the heat exchanger.

ii. To build a suitable heat exchanger to cool and heat the blood.

iii. To design and analyze the heat exchanger.

iv. Achieve a lower pressure drop, thereby achieving a low priming volume.

v. Easy to clean and sterilize the heat exchanger.

vi. Greater heat transfer rate of the heat exchanger.

4. METHODOLOGY AND CALCULATION

METHODOLOGY As the objective of this present project is to design suitable heat exchanger to cool and heat the blood for the required temperature.

- Collect all required parameters about blood, water, and heat exchanger dimensions.
- Select a suitable pump to flow the blood from the human body to the heat exchanger.
- Calculate the fluid velocity of both fluids.
- Calculate the Reynolds number of both fluid flow, based on the Reynolds number (<2100 laminar flow, >2100 turbulent flow) we can choose heat transfer coefficient formula.
- Further, calculate the heat removed (heat duty) from the blood and water.
- The main objective in the design of a heat exchanger is to determine the surface area required for the specified duty of both fluids using the temperature differences available.
- Calculate the pressure drop of both fluids.

5. CALCULATIONS

On the basis of the Project objectives, we will be calculating the values of the basic dimensions of the shell and tube heat exchanger. The thermal design will only be considered. The Edward S. Gaddis et al method will be employed for the calculation.

Mathematical modelling involves calculating unknown temperatures by using input parameters.

The detailed procedure is given below.

Pump Specifications: - The BP-50 Bio-Pump Centrifugal Blood Pump is used in paediatric cardiac surgery.

- 6.35 mm inlet and outlet ports
- Flow range 0 to 1.5 litters/minute

Many Cardio Pulmonary Bypass circuits now employ a centrifugal pump for the maintenance and control of blood flow during cardiac surgery. By altering the speed of revolution (RPM) of the pump head, blood flow is produced by centrifugal force.

Input data

Hot Fluid = Blood

Cold Fluid = Water

Inlet Temperature of hot fluid $T_{hi} = 37\ °C$

Outlet Temperature of hot fluid $T_{ho} = 27\ °C$

Inlet Temperature of cold fluid $T_{ci} = 10\ °C$

Outlet Temperature of cold fluid T_{co} =? °C
Mass –flow rate of hot fluid M_h = 0.025 kg/s
Mass –flow rate of cold fluid M_c = 0.2 kg/s
Specific Heat of hot fluid C_{ph} = 3330 J/kg K
Specific Heat of cold fluid C_{pc} = 4182 J/kg K
Thermal conductivity of hot fluid K_h = 0.426 W/m K
Thermal conductivity of cold fluid K_c = 0.57646 W/m K
Density of hot fluid ρ_h = 1060 Kg/m^3
Density of cold fluid ρ_c = 999.64 Kg/m^3
Dynamic viscosity of hot fluid μ_h = 0.0035 Kg/m s
Dynamic viscosity of cold fluid μ_c = 0.001285 kg/m s
Number of tubes n_t = 19
Length of tube L = 0.280 m
Outer diameter of tube d_o = 0.0048 m
Inner diameter of tube d_i = 0.0041 m
Outer diameter of shell D_i = 0.043 m
Inner diameter of shell D_o = 0.040 m

Material selected = Stainless Steel 304
Thermal conductivity of Material K_m = 16.2 W/m K

Tube side calculation

Mean Temperature of the hot fluid

$$T_{hmean} = \frac{T_{hi} + T_{ho}}{2} = \frac{37+27}{2}$$

$$T_{hmean} = 32^{o}C$$

The volumetric flow rate of the hot fluid

$$V_{fh} = \frac{M_h}{\rho_h} = \frac{0.025}{1060}$$

$$V_{fh} = 0.000023584 \ m^3/s$$

Prandtl number of hot fluid

$$P_{rh} = \frac{\mu_h \times C_{ph}}{K_h} = \frac{0.0035 \times 3330}{0.426}$$

$$P_{rh} = 27.36$$

The cross-sectional flow area of tubes

$$a_t = \left(\pi \times \frac{d_i^{\,2}}{4}\right) \times N_t = \left(\pi \times \frac{0.0041^2}{4}\right) \times 19$$

$$a_t = 0.000250848 \ m^2$$

The velocity of the hot fluid

$$V_h = \frac{M_h}{\rho_h \times a_t} = \frac{0.025}{1060 \times 0.00025088}$$

$$V_h = 0.094 \ m/s$$

Reynolds number of hot fluid

$$R_{eh} = \frac{\rho_h \times V_h \times d_i}{\mu_h} = \frac{1060 \times 0.094 \times 0.0041}{0.0035}$$

$$R_{eh} = 116.732$$

Tube side Heat transfer coefficient

$$h_i = 1.86 \times \frac{K_h}{d_i} \times (R_{eh} \times P_{rh})^{0.33} \times \left(\frac{d_i}{L}\right)^{0.33}$$

$$= 1.86 \times \frac{0.426}{0.0041} \times (116.732 \times 27.36)^{0.33} \times \left(\frac{0.0041}{0.280}\right)^{0.33}$$

$$h_i = 687.41 \ W/m^2K$$

The heat removed from the hot fluid

$$Q_h = M_h \times C_{ph} \times (T_{inlet} - T_{outlet})$$

$$= 0.025 \times 3330 \times (37 - 27)$$

$$Q_h = 832.5 \ W$$

Provided Heat transfer surface area

$$A_P = \pi \times d_o \times L \times n_t$$

$$= \pi \times 0.0048 \times 0.280 \times 19$$

$$A_P = 0.0802 \ m^2$$

Calculation of pressure drop at the tube side

The friction factor of tube

$$\xi_t = 0.0056 + 0.5 \times R_{eh}^{-0.32}$$

$$= 0.0056 + 0.5 \times 116.732^{-0.32}$$

$$\xi_t = 0.1146$$

Pressure drop at the tube side

$$\Delta P = \xi_t \times \frac{\rho_h \times V_h^2}{2 \times d_i} \times \Delta L$$

$$= 0.1146 \times \frac{1060 \times 0.094^2}{2 \times 0.0041} \times 0.280$$

$$\Delta P = 36.66 \ Pa$$

Shell side calculation

Consider energy balance equation to find out unknown values of temperatures. Consider some input parameters like inlet temperature of hot fluid and cold fluid.

The energy balance equation is given as

Energy Balance Equation

$$M_c \times C_{pc} \times (T_{co} - T_{ci}) = M_h \times C_{ph} \times (T_{hi} - T_{ho})$$

$$0.2 \times 4182 \times (T_{co} - 10) = 0.025 \times 3330 \times (37 - 27)$$

$$836.4 \times (T_{co} - 10) = 832.5$$

$$= \frac{835.5}{836.4} + 10$$

Outlet Temperature of cold fluid T_{co} = 10.99534°C

Mean Temperature of cold fluid

$$T_{c\ mean} = \frac{T_{ci} + T_{co}}{2}$$

$$= \frac{10 + 10.99534}{2}$$

$$T_{c\ mean} = 10.498°C$$

Heat removed from cold fluid

$$Q = M_c \times C_{pc} \times (T_{co} - T_{ci})$$

$$= 0.2 \times 4182 \times (10.99534 - 10)$$

$$Q = 832.5\ W$$

Log Mean Temperature Difference, LMTD for Counter Flow

$$\Delta T_1 = T_{hi} - T_{co} = 37 - 10.99534 \quad \Delta T_1 = 26.00466°C$$

$$\Delta T_2 = T_{ho} - T_{ci} = 27 - 10 \quad \Delta T_2 = 17°C$$

$$LMTD, \Delta T_m = \frac{\Delta T_1 - \Delta T_2}{\ln\left(\frac{\Delta T_1}{\Delta T_2}\right)} = \frac{26.00466 - 17}{\ln\left(\frac{26.00466}{17}\right)}$$

$$\Delta T_m = 21.1843°C$$

Overall heat transfer coefficient in Shell and tube heat exchanger,

$$U = \frac{1}{\frac{d_o}{d_i} \times \frac{1}{h_i} + \frac{d_o \times \ln(d_o / d_i)}{2 \times K_{material}} + \frac{1}{h_o}}$$

$$= \frac{1}{\frac{0.0048}{0.0041} \times \frac{1}{687.41} + \frac{0.0048 \times \ln(0.0048/0.0041)}{2 \times 16.2} + \frac{1}{4276.24}}$$

$$U = 510.11\ W/m^2k$$

Required Heat Transfer Surface Area

$$A_R = \frac{Q}{U \times Lmtd} = \frac{832.5}{510.23 \times 21.1843}$$

$$A_R = 0.0770\ m^2$$

Calculation of pressure drop at Shell side

Calculation of the pressure drop ΔP_Q in a central cross flow section:

Mean Wall Temperature

$$T_{Mean\ wall} = \frac{T_{h\ mean} + T_{c\ mean}}{2} = \frac{10.498 + 32}{2}$$

$$T_{mean\ wall} = 21.249\ °C$$

Pressure drop in a central cross flow section two adjacent baffles

$$\Delta P_Q = \Delta P_{Q,0} \times f_L \times f_B = 91.7358 \times 0.3839 \times 0.396531$$

$$\Delta P_Q = 13.9657\ Pa$$

Pressure drop in an end cross flow section

$$\Delta P_{QE} = \Delta P_{QE,0} \times f_B = 122.3144 \times 0.396531$$

$$\Delta P_{QE} = 48.5014\ Pa$$

Pressure drop in window section

$$\Delta P_W = \sqrt{\Delta P_{W,lam}^2 + \Delta P_{w,turb}^2} \times f_{z,t} \times f_L$$

$$= \sqrt{347.8599^2 + 440.8272^2} \times 0.96129 \times 0.3839$$

$$\Delta P_W = 207.2886\ Pa$$

Pressure drop in both inlet and outlet nozzles

$$\Delta P_N = \xi_N \times \frac{\rho_C \times W_N^2}{2} = 1.6195 \times \frac{999.64 \times 2.5464^2}{2}$$

$$\Delta P_N = 5247.79484\ Pa$$

Shell side pressure drop

$$\Delta P = (n_B - 1) \times \Delta P_Q + 2 \times \Delta P_{QE} + n_B \times \Delta P_W + \Delta P_N$$

$$= (7-1) \times 13.9657 + 2 \times 48.5015 + 7 \times 207.2886 + 5247.79484$$

$$\Delta P = 6879.612647\ Pa$$

6. RESULTS & DISCUSSIONS

Analytical Results

Table 1: Analytical result of hot fluid

Hot Fluid				
Mass flow rate Kg/s	Temperature °C		Velocity m/s	Pressure drop Pa
	Inlet	Outlet		
0.025	37	27	0.094	36.66

Table 2: Analytical result of cold fluid

Cold Fluid				
Mass flow rate Kg/s	Temperature °C		Velocity m/s	Pressure drop Pa
	Inlet	Outlet		
0.2	10	10.99	2.54	6879.61

In beginning of the project we calculated the design values for our heat exchanger and calculated the outlet temperature of cold fluid, velocity and pressure drop of both hot and cold fluids. Those calculated values were used in this CFD simulation for the analysis of our heat exchanger.

7. CFD ANALYSIS OF SHELL AND TUBE TYPE HEAT EXCHANGER

INTRODUCTION TO COMPUTATIONAL FLUID DYNAMICS

Computational fluid dynamics (CFD) is a computer-based simulation method for analysing fluid flow, heat transfer. This project uses CFD for analysis of flow and heat transfer. ANSYS FLUENT is the commercial CFD package used to solve the governing integral equations of conservations of mass, momentum and energy and for other scalars like turbulence and chemical species. In all cases it uses control volume technique, which consists of;

- Using the computational grid in which the domain is divided into discrete control volumes.
- Integration of governing equations on individual control volumes to construct algebraic equations for discrete dependent variables (unknowns), such as velocities, pressure, temperature, and conserved scalars.
- Linearization of discretized equations and solution of resultant linear equation system, to yield updated values of dependent variables.

GEOMETRY MODELING

The flow volume of the geometry has been created by using Design Modeller and mesh is generated in Ansys fluent and the analysis carried out in Ansys Fluent Solver. A software model is accomplished by using proportions of shell, tubes and baffles in ANSYS 16. Workbench design modeler is used to construct heat exchanger geometry and for further analysis. This heat exchanger is counter flow type. The geometry clean-up is done in the design modeler, it include tools to repair, prepare and optimize models. The tube side consists of one inlet and one outlet representing of 19 tubes and shell side also one inlet and one outlet is present. After model is generated, run thermal simulation of heat exchanger. The outcomes prevailed were moderately well known with general circumstance. The parts independently as well as in congregation are as shown below Fig

SPECIFICATIONS

1. Shell inner dia. = 40 mm
2. Shell outer dia. = 43 mm
3. Length = 280 mm
4. Tube inner dia. = 4.1 mm
5. Tube outer dia. = 4.8 mm
6. Tube length = 280 mm
7. Tube material = Stainless steel
8. Shell material = Stainless steel
9. Tube type = Staggered pitch
10. Pitch length = 7.4 mm
11. Total number of the tubes = 19
12. Fluid for shell side = Water
13. Fluid for tube side = Blood

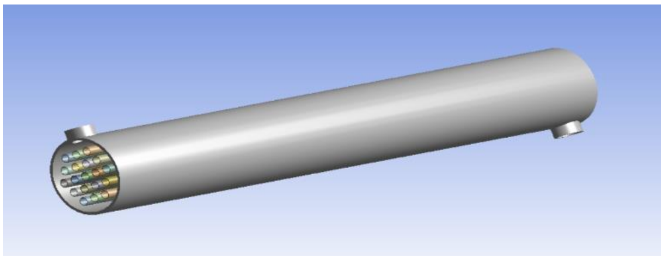

Fig 1: Shell and tube heat exchanger (isometric view)

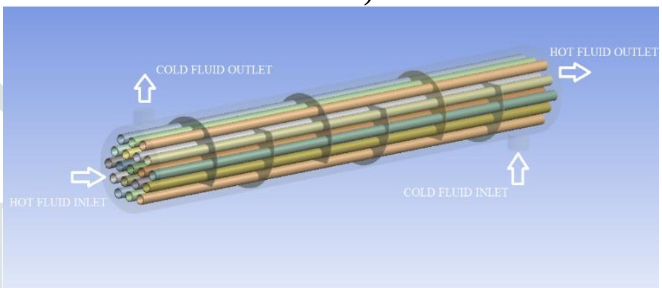

Fig 2: Transparent view of shell and tube heat exchanger

MESH GENERATION A pre-processing step for the computational field simulation is the discretization of the domain of interest and is called mesh generation. The process of mesh generation can be broadly classified into two categories based on the topology of the elements that fill the domain. These two basic categories are known as structured and unstructured meshes. The different types of meshes have their advantages and disadvantages in terms of both solution accuracy and the complexity of the mesh generation process. A structured mesh is defined as a set of hexahedral elements with an implicit connectivity of the points in the mesh. The structured mesh generation for complex geometries is a time-consuming task due to the possible need of breaking the domain manually into several blocks depending on the nature of the geometry. An unstructured mesh is defined as a set of elements, commonly tetrahedrons, with an explicitly defined connectivity. The unstructured mesh generation process involves two basic steps: point creation and definition of connectivity between these points. Flexibility and automation make the unstructured mesh a favourable choice although solution accuracy may be relatively unfavourable compared to the structured mesh due to the presence of skewed elements in sensitive regions like boundary layers.

MESHING OF COLD FLUID VOLUME

The Cold fluid flow volume of the geometry has been created by using Ansys design modeler and mesh is generated in Ansys fluent and the analysis carried out in Ansys Fluent Solver.

In the present study, unstructured grid i.e. Advance size function has been used since it will improve mesh quality and creates proximity and curvature and at the hole side and narrow sections. The geometry clean-up is done in the Ansys design modeler, it include tools to repair, and optimize models. Volume extraction has been done in Ansys design modeler. Fluent meshing has the capabilities of generate CFD meshes on highly complex & dirty geometry and it contains tools to handle, repair & improve boundary meshes. Fine mesh has been generated around the boundary layer of shell, tube side and baffle side. The following table gives details of generated grid. Around 4.1 million grid size has been employed. Aspect ratio and orthogonal quality are maintained well within the limit.

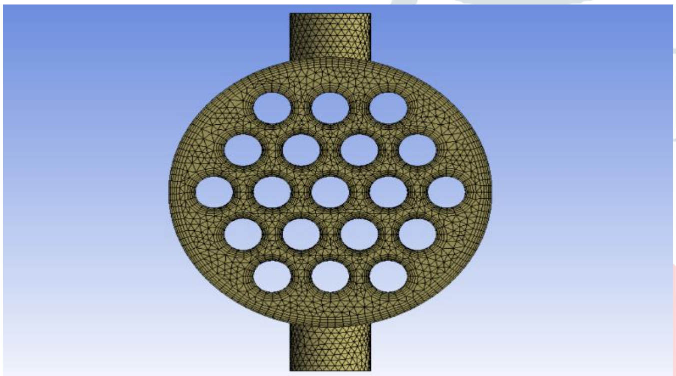

Fig 3: Cold Fluid Domain Mesh

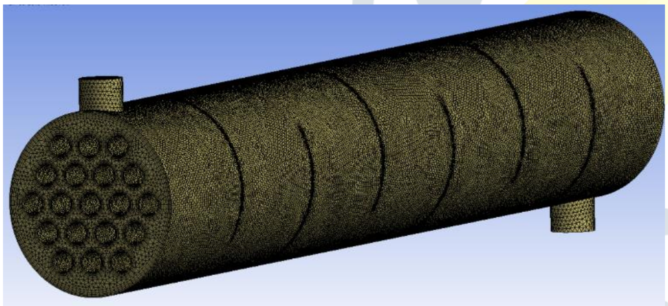

Fig 4: Isometric view of Cold Fluid Domain Mesh

GRID GENERATION DETAILS:

Table 3: Grid generation details

Elements	4.1 Million
Skewness	0.89
Minimum Orthogonal Quality	0.28
Maximum Aspect Ratio	3.53326e+01

MESHING OF TUBES

The Tubes of the geometry has been created by using Ansys Design Modeller and mesh is generated in Ansys fluent. An advanced size function is used, approximately 0.49 million cells has been generated. While making the grid, it has been estimated that quality of grid in terms of Aspect ratio and orthogonality is good.

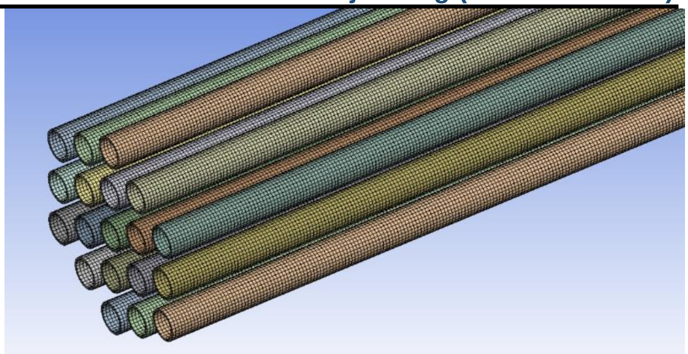

Fig 5: Mesh of Tubes

Table 4: Grid generation details

Elements	0.49 Million
Skewness	0.81
Minimum Orthogonal Quality	0.51
Maximum Aspect Ratio	2.14326e+01

MESHING OF HOT FLUID VOLUME

The Hot fluid flow volume of the geometry has been created by using Ansys Design Modeller and hot fluid volume mesh is generated in Ansys fluent and the analysis carried out in Ansys Fluent Solver. In the present study, structured grid i.e. Advance size function has been used since it will improve mesh quality. Volume extraction has been done in Ansys design modeler. Fine mesh has been generated around the boundary layer of tube side. The following table gives details of generated grid. Around 1.1 million grid size has been employed. Aspect ratio and orthogonal quality are maintained well within the limit.

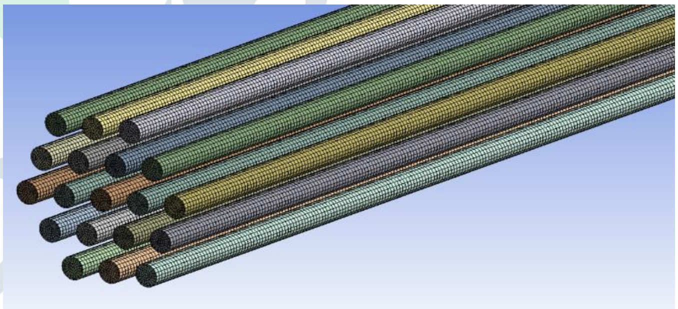

Fig 6: Hot Fluid Domain mesh

Table 5: Grid generation details

Elements	1.1 Million
Skewness	0.74
Minimum Orthogonal Quality	0.56
Maximum Aspect Ratio	7.5

Boundary condition

The boundary conditions specified for the systems under study are given in the table 1. The solver details used while carrying out analysis using Ansys fluent are mentioned below ANSYS FLUENT 16.0 is used for the analysis. Studies are carried out considering the flow to be steady, incompressible and laminar flow of both fluids. Pressure based solver – SIMPLE is used for solving the governing equations. Solution is first order accurate for all equations. Laminar is used for modelling laminar flow.

Table 6: Boundary conditions

S.NO	SOLUTION SETUP	INPUTS
1.	General	3D, Pressure based solver, Absolute Velocity formulation, Steady state
2.	Model	Energy Viscous – Laminar
3.	Materials	Blood Water Stainless Steel
4.	Cell Zone Conditions	Hot Fluid – Blood Cold Fluid – Water Tubes – Stainless Steel
5.	Boundary Condition	Hot Fluid inlet – Mass flow rate and Temperature Cold Fluid inlet – Mass flow rate and Temperature
6.	Solution Method	Formulation – SIMPLE scheme Gradient – least square cell based Pressure – Standard Momentum – First order upwind Energy – First order upwind

Based on the above conditions in the domain, simulations where carried out in Ansys Fluent and the obtained solutions are as discussed in next chapter.

INLET BOUNDARY CONDITIONS:

INLET:

Table 7: Inlet Boundary Condition for Hot Fluid

Entities	
Inlet Mass flow rate	0.025 Kg/s
Inlet Temperature	37 °C

INLET:

Table 8: Inlet Boundary Condition for Cold Fluid

Entities	
Inlet Mass flow rate	0.2 Kg/s
Inlet Temperature	10 °C

SOLVER CONDITIONS In the present analysis, pressure based solver is used since the flow is incompressible. For pressure velocity- coupling (SIMPLE scheme) is used with pressure as standard and first order upwind scheme is used for momentum, turbulent kinetic energy for discretisation of terms.

8. CFD RESULTS AND DISCUSSIONS

CFD RESULTS

The outlet temperature of hot fluid and cold fluids have been analysed. The velocity and pressure drop of both fluids is also calculated. Heat transfer and flow distribution is discussed in detail and the results obtained are tabulated as below.

Table 9: CFD results for hot fluid

Hot Fluid					
Mass flow rate Kg/s		Temperature °C		Velocity m/s	Pressure drop Pa
Inlet	Outlet	Inlet	Outlet		
0.025	-0.025	37	28.25	0.095	36.86

Table 10: CFD results for cold fluid

Cold Fluid					
Mass flow rate Kg/s		Temperature °C		Velocity m/s	Pressure drop Pa
Inlet	Outlet	Inlet	Outlet		
0.20	-0.20	10	11.18	2.58	6880.17

The analysis results are carried out in components wise such as shell and tube and under counter flow conditions. Water is used as cold fluid at the shell side and blood is used as hot fluid at the tube side. Boundary conditions used are cold fluid inlet mass flow rate and temperature and hot fluid mass flow rate and temperature. Exit mass flow rate was monitored for the convergence.

HOT FLUID TEMPERATURE CONTOUR:

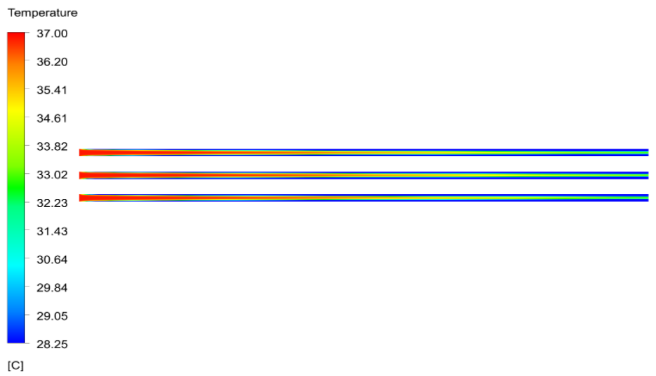

Fig 7: Temperature distribution in hot fluid

From Fig 7 it shows Temperature distribution in the hot fluid. The temperature gradually decreasing from 37 °C to 28.25 °C. The objective of the project is to reduce the hot fluid temperature, we can clearly see at a temperature of hot fluid has significantly decreased at the outlet of the tube when compared to the temperature at the inlet. As the cold fluid is flowing around the hot fluid, the cold fluid gains the temperature from the hot fluid so that the temperature of the cold fluid increases. Due to a loss in heat, the temperature of the hot fluid at the outlet decreases.

COLD FLUID TEMPERATURE CONTOUR:

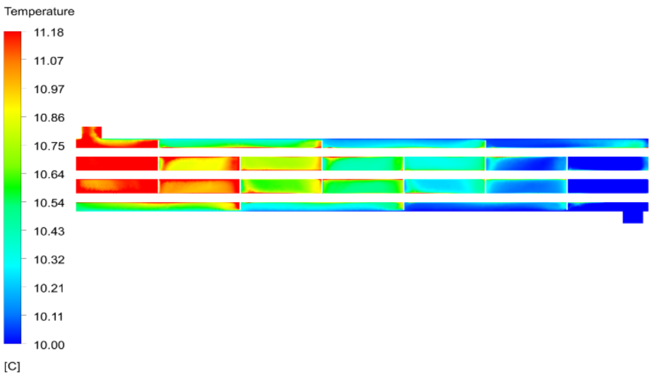

Fig 8: Temperature distribution in cold fluid

Fig 8 shows the temperature distribution in cold fluid. The temperature gradually increases from 10 °C to 11.18 °C. The objective of the project is to reduce the hot fluid temperature, for that we are using cold fluid. We can clearly see the temperature of cold fluid has significantly increased at the outlet of the shell when compared to the temperature at the inlet. As the hot fluid is flowing inside the tube, the cold fluid gains the temperature from the hot fluid so that the temperature of the cold fluid increases. Due to a loss in heat, the temperature of hot fluid decreases.

HOT FLUID VELOCITY CONTOUR:

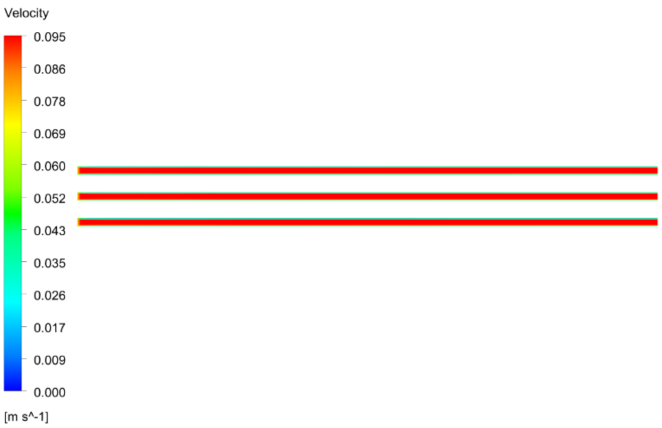

Fig 9: Variation of velocity in hot fluid

Fig 9 shows the variation of velocity in the hot fluid. Based on the Reynolds number the flow is laminar. Laminar flow is characterized by the smooth flow of the fluid in layers that do not mix. Velocity is different in each layer, which we can clearly observe that the variation of velocity in hot fluid does not change too much. Due to some wall friction velocity decreases at the wall side and in the middle of the fluid, velocity and also the mass flow rate of the fluid remains unchanged. Since the cross-section of the flow remains the same, the velocity of the fluid throughout the flow will remain constant.

COLD FLUID VELOCITY CONTOUR:

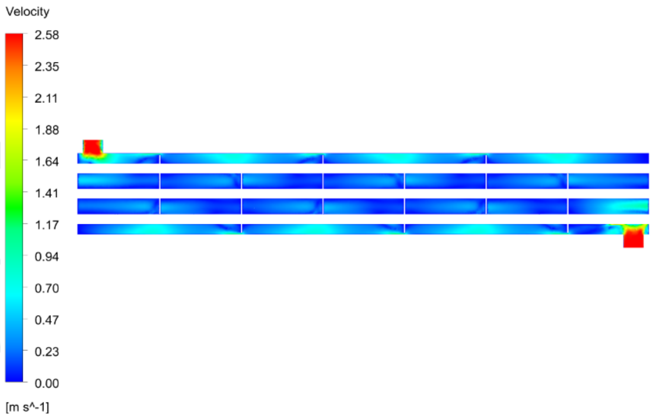

Fig 10: Variation of velocity in Cold fluid

Fig 10 shows the variation of velocity in cold fluid with respect to the cross-sectional area of the shell with baffles. The velocity of the cold fluid changes along the cross-section. We can notice the maximum velocity of cold fluid will be at the inlet and outlet of the shell and minimum velocity at the baffle side. Due to wall friction and baffles, the velocity of the cold fluid continuously changes along with the length of the shell.

HOT FLUID PRESSURE DROP CONTOUR:

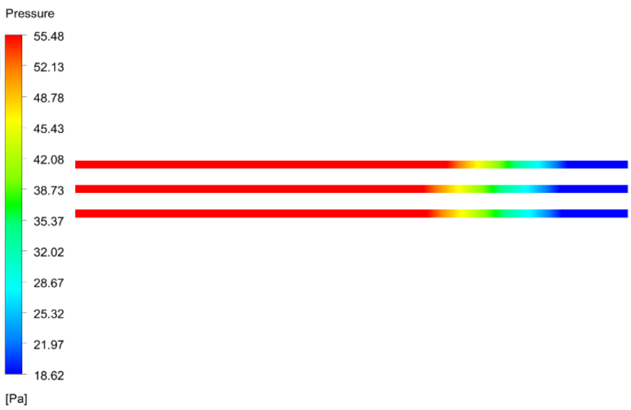

Fig 11: Variation of pressure in hot fluid

Fig 11 shows the variation of pressure drop in hot fluid at the tube side. When fluid flows through a tube there will be a pressure drop that occurs as a result of resistance to flow. A pressure drop occurs when frictional forces, caused by the resistance to flow, act on a fluid as it flows through the tube. We can clearly notice the pressure variation in tube along the length. Pressure drop is very less in horizontal tubes.

COLD FLUID PRESSURE DROP CONTOUR:

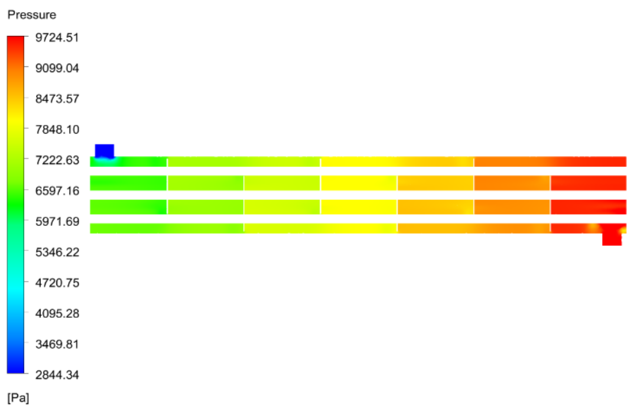

Fig 12: Variation of pressure in cold fluid

Fig 12 shows the variation of pressure drop in cold fluid at the shell side. Due to baffles and different cross-sectional areas, the pressure drop is more at the shell side. From the fig we can understand pressure drop occurs more at the baffle side.

9. CONCLUSION

In this project, the thermal design of a heat exchanger for cooling blood during open-heart surgery has been presented. The heat transfer and flow distribution are discussed in detail and the proposed model is compared with analytical results. The model predicts the heat transfer, velocity and pressure drop with an average error of 2%. Thus the model can be improved. The blood heat exchanger is widely used in open-heart surgery for cool and rewarm the blood. Today the main problem of the heat exchanger is not reusable, bulky in size and having a high priming volume, more pressure drop, and less heat transfer rate. The following conclusions are drawn from the project.

- A new type of heat exchanger suited to cool the blood during open heart surgery profound is described. It is distinguished by a small priming volume and a high heat transfer rate. It is of 'shell and tube type' and features counter flow, consisting of tubes which can be cleaned and sterilized easily.
- Centrifugal pump is used to pump the blood through the heat exchanger from human body. The pump outlet mass flow is considered as the inlet mass flow rate for the heat exchanger.
- The diameter of tube influences the heat transfer characteristics. Since the surface area of smaller tubes is less, the heat transfer rate is also lower. Hence a suitable diameter has been chosen to increases the heat transfer rate.
- We can reuse the heat exchanger after steam sterilization with 132 °C. This temperature must be maintained for a minimal time to kill microorganisms.
- From the CFD analysis, it is found that the hot fluid temperature decreases with increase in cold fluid temperature. The velocity and pressure drop of the hot fluid and cold fluid does not vary much as compared to analytical results.
- ANSYS and Analytical results are compared and found in good agreement, thus proving the strength of the model. After completing CFD analysis results, we can say that CFD Analysis is a good tool to avoid costly and time-consuming Experimental work.

REFERENCE

1. *Harrison, "Blood heat exchanger", September 9, 1980.*
2. *Swan, "Hypothermia for open heart surgery" 1958.*
3. *Vasulin and oslejsek, "Heat exchanger for profound hypothermia" 1965.*
4. *Ahuja, and Hendee, "Thermal design of a heat exchanger for heating or cooling blood", Vol 23, 1978.*
5. *Russell, "A Discussion of the problems of the heat exchange blood warming devices" 1969.*
6. *Shore, "Heat exchange in profound hypothermia: heat exchanger design for blood during external circulation", Vol 171, 1963.*
7. *Edward S. Gaddis and Volker Gnielinski, "VDI heat atlas second edition", vol G8 731-741 and 1095 -1105, 2010*
8. *Dr. Reyad Shawabkeh, "Steps for design of Heat Exchanger" vol 6, 1985*
9. *Sadick kalac, Hongtan Liu and Anchasa Pramuanjaroenkij "Heat Exchanger; Selection, Rating, and Thermal Design Third Edition" page 455 to 513, 2012*