

ANALYSIS OF TUBULAR HEAT EXCHANGER MODELS IN ANSYS FLUENT

A Dissertation submitted to the Delhi Technological University, Delhi in fulfillment of the requirements for the award of the degree

Master in Technology
In
Mechanical (Thermal) Engineering
By
UJJAWAL KUMAR
(2K19/THE/12)

Under the supervision of
Dr. NAUSHAD AHMAD ANSARI



Mechanical Engineering Department

Delhi Technological University

Shahbad Daulapur Bawana Road

Delhi – 110042, India

June 2022

DELHI TECHNOLOGICAL UNIVERSITY
(Formerly Delhi College of Engineering)
Bawana Road, Delhi - 110042

CANDIDATE'S DECLARATION

I, UJJWAL KUMAR Roll No. 2K19/THE/12 student of M.Tech (Thermal Engineering), hereby declare that the project Dissertation titled "**ANALYSIS OF TUBULAR HEAT EXCHANGER MODELS IN ANSYS FLUENT**" which is submitted by me to the Department of Mechanical Engineering, Delhi Technological University, Delhi in partial fulfilment of the requirement for the award of the degree of Master of Technology is original and not copied from any source without proper citation. This work has not previously formed the basis for the award of any degree, Diploma Associateship, fellowship, or other similar title or recognition.

Place: Delhi

Date: 10/06/2022

Ujjwal Kumar
14/06/2022

UJJWAL KUMAR

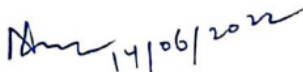
Roll No. 2K19/THE/12

DELHI TECHNOLOGICAL UNIVERSITY
(Formerly Delhi College of Engineering)
Bawana Road, Delhi -110042

CERTIFICATE

This is to certify that Project Dissertation entitled "**ANALYSIS OF TUBULAR HEAT EXCHANGER MODELS IN ANSYS FLUENT**" which is submitted by UJJWAL KUMAR, RollNo.2K19/THE/12 Department of Mechanical Engineering, Delhi Technological University, Delhi in partial fulfilment of the requirement for the award of Degree of Master of Technology is the record of the project work carried out by the student under our supervision. To the best of our knowledge this work has not been submitted in part or full for any Degree or Diploma to this University or elsewhere.

PROJECT GUIDE

14/06/2022

DR. NAUSHAD AHMAD ANSARI

Assistant Professor

Date: 10/06/2022

Place: Delhi

ACKNOWLEDGEMENT

I, UJJWAL KUMAR, take this momentous opportunity to express my heartfelt gratitude and highly esteemed main guide DR. NAUSHAD AHMAD ANSARI, Assistant Professor, Department of Mechanical Engineering, Delhi Technological University for providing us an opportunity to present our project on "**ANALYSIS OF TUBULAR HEAT EXCHANGER MODELS IN ANSYS FLUENT**". I with full pleasure convey our heartiest thanks to Professor SK Garg, HOD, Department of Mechanical Engineering, Delhi Technological University for the invaluable advice and whole hearted cooperation without which this project would not have seen the light of day. I attribute the heartiest thanks to all the faculty of the Department of Mechanical Engineering and friends for their valuable advice and encouragement.

UJJWAL KUMAR

(2K19/THE/12)

Department of Mechanical Engineering,

Delhi Technological University

ANALYSIS OF TUBULAR HEAT EXCHANGER MODELS IN ANSYS FLUENT

ABSTRACT

The heat exchanger is an essential component of any system in terms of heat exchange and energy conservation. Cross flow and counter flow heat exchangers are utilized in a variety of industries, including the oil and petrochemical industries, cooling, food storage, and others. They are employed because they produce a wide variety of outputs, are simple to set up, and have large maintenance expenses. Shell and tube heat exchangers are the most commonly used type of heat exchanger when doing a lot of chemical work, such as creating oil. Tubular heat exchangers, on the other hand, may take on a variety of shapes, including ellipses, rectangles, and even round/flat twisted tubes. By adjusting the tube diameter, length, and arrangement, it is possible to readily alter the core form. When it comes to design, there is a lot of freedom for imagination to flourish. Tubular exchangers are capable of handling high external pressures as well as large fluctuations in fluid pressure levels. Tubular heat exchangers are often used for the transmission of heat from one liquid to another and from one liquid to another phase change (condensing or evaporating). Using the CFD application, the simulation strategy is to display and cross-section the main geometry of shell and cylinder heat exchangers. ANSYS 14.0 is a simulation program. ANSYS programming tools will be used to create a shell and tube heat exchanger with a spiral-shaped baffle. This will assist in keeping the flow and temperature of the water and air inside the shell. In this study, there are three types of heat exchangers, each with cylinders that are 20 mm wide and 1000 mm long, with a shell width of 220 cm. Each is made up of 20 mm broad cylinders. A study is conducted to determine how much heat may be exchanged in order to determine the ideal specifications for heat exchangers. Because the temperatures of the shell and cylinder heat exchangers differ, the yield parameter is the temperature differential between them. To obtain more information, the prefix parameter (tube distance across) is used. Based on tubular heat exchangers, three types of models are created, and CFD analysis is performed in ANSYS 14.0. So, the best section of a heat exchanger for regulating water temperature may be identified, and this is how it works.

***Keywords* - Shell and tube heat exchanger, FEM, ANSYS, Thermal Analysis.**

TABLE OF CONTENTS

S.NO	LIST OF CONTENT	PAGE NO
CHAPTER 1	INTRODUCTION	01
1.1	Introduction	02
1.2	Heat Exchanger	03
CHAPTER 2	LITERATURE REVIEW	12
2.1	Literature Review	13
2.2	Conclusion	20
CHAPTER 3	FINITE ELEMENT METHOD	21
3.1	Introduction to FEM	22
3.2	General procedure of finite element method	22
3.3	Convergence requirement	24
3.4	Advantages of FEM	24
3.5	Applications of FEM	25
3.6	FEA SOFTWARE – ANSYS	26
3.7	Evolution of ANSYS Program	26
3.8	Outline of the program	26
3.9	Costs of ANSYS conception and development are reduced	28
3.10	Material properties	28
3.11	Obtain the solution	29
3.12	Pre – Processor	29
CHAPTER 4	METHODOLOY AND MODELLING	32
4.1	Introduction	33

4.2	Computational Fluid Dynamics	33
4.3	Basic Structure of A CFD Code	34
4.4	ANSYS CFX (Fluid flow) Workbench Environment	36
4.5	Create a Geometry	37
4.6	Meshing of Geometry (discretization of the computational domain)	40
CHAPTER 5	IMPLEMENTATION	43
5.1	Introduction	44
5.2	Factors Taken for Result	44
5.3	Shell and Tube heat exchanger designing in ANSYS	45
5.4	Set up of ANSYS software	48
CHAPTER 6	Results and Discussion	55
6.1	Introduction	56
6.2	Results of Heat Exchanger	56
6.3	Velocity of Tubes at outlet	63
6.4	Pressure of Tubes at outlet	66
Chapter 7	Conclusion	69
7.1	Conclusion	69
7.2	Future Scope	70
	References	71

LIST OF FIGURES

FIGURE NAME	LIST OF FIGURE	PAGE NO
1.1	A Shell and Tube heat exchanger	03
1.2	Countercurrent (A) and parallel (B) Flow	03
1.3	Tube bundle and insertion	05
1.4	C Types of pitch layout	06
1.5	Plates of heat exchanger.	07
1.6	single plate of heat exchanger	07
1.7	The reboiler kettle in an industrial distillation tower	08
1.8	Helical coil type pipe	10
1.9	Helical Pipe Heat Exchanger	11
4.1	ANSYS workbench (CFD)	36
4.2	Heat Exchanger with Parallel Tubes	38
4.3	Heat Exchanger with 'U' Pattern type flow Tube	39
4.4	Helical Flow Tube Heat Exchanger	39
4.5	Zigzag tube setup in heat exchanger model	40
4.6	Meshed model of heat exchanger	41
4.7	Details of meshing, CFD simulations of Heat Exchanger model	42
5.1	Straight tubes used inside the heat exchanger model	46
5.2	'S' Pattern tube used inside heat exchanger model	47
5.3	Helical type Tube used inside Heat Exchanger	47
5.4	Zigzag type Tube used inside heat exchanger model	48
6.1	Temperature fluctuations in parallel pipe flows	56
6.2	Temperature changes in the 'S' Pattern pipe flow	57
6.3	temperature fluctuations in helical pipe flow	57

6.4	Pipe flow temperature changes in a zigzag pattern	58
6.5	Temperatures of cold water in tube heat exchangers	59
6.6	Temperature of hot water from the shell at the parallel tube's output	60
6.7	Temperature of hot water from the shell at the 'S' Pattern tube's output	60
6.8	Temperature of hot water from the shell at the tube's outflow in a helical pattern	61
6.9	Temperature of hot water from the shell at the tube's outflow in a zigzag pattern	61
6.10	Hot water temperatures in heat exchangers dependent on tube layout	62
6.11	Flow rate of a parallel tube pipe Streamline	63
6.12	Tube pipe flow velocity Streamline 'S' Pattern	64
6.13	streamline velocity flow in a helical tube and a shell	64
6.14	The zigzag tube and shell combine to create a high-velocity flow.	65
6.15	Variations of Velocity at outlets of tubes	66
6.16	Solar water model designed in Solidworks	68
6.17	Solar water model in Ansys	69
6.18	Temperature at outlet in Solar water model	69

LIST OF TABLES

TABLE NAME	LIST OF TABLE	PAGE NO
4.1	Dimensions and materials of a shell and tube heat exchanger	37
5.1	heat exchanger 3d model dimensions	47
6.1	Temperatures of cold-water flow at the intake and outflow	58
6.2	Temperatures of hot water at the inflow and outflow	62
6.3	Comparison of Tube Velocity at Outlet	63

CHAPTER 1
INTRODUCTION

CHAPTER 1

INTRODUCTION

1.1. Introduction

When two liquids come into direct touch with each other, they exchange heat. Open water radiators and steam condensers, such as the ones seen below, are two common examples of this style of design. It is a heat exchanger with a partition that separates the liquids. The splitter can be simple and flat, or it can include several fins, baffles, and tubes, depending on how difficult you want it to be. In a heat exchanger, one stream of hot liquid comes into touch with another stream of cold liquid. This is done to allow more heat to be transmitted from the hot to the cold liquid stream. It has a good quantity of surface area for heat exchange for a given amount of equipment. Heat exchangers are used in a variety of industries, including compound process firms, control generation, waste heat recovery, cryogenic, cooling, petrochemical enterprises, and so on.

It is made up of a shell and a huge pressure tank that holds many cylinders. When one liquid runs through a group of cylinders and another flows over the cylinders, this is demonstrated. Heat is shared between the two types of liquid in this way. Heat exchange measurement is used to determine how well and efficiently processes are done. Direct contact heat exchangers and indirect contact heat exchangers are the two types of heat exchangers that work with heat. They make form a shell and tube heat exchanger. This is a heat exchanger that does not come into direct contact with water. Heat exchange may be improved using both dynamic and non-involved ways. Surface vibration infusion, electrostatic fields, and other techniques are included in a dynamic approach [1], enlarged surfaces, and other mechanical alterations], as well as other solutions, are examples of active techniques. Baffles are used to move the liquid in the shell and to direct the flow of liquid in each cylinder. Using this ratio, you may determine how far apart one baffle is from the next. Turbulence is generated by a split shell and cylinder heat exchanger, which allows heat to circulate more easily. By increasing, the power of disturbance level stream opposition can be increased, and significant weight loss can be obtained [3]. To make matters worse, the intensity is now being employed more frequently. This is a significant problem. Heat exchangers that generate more noise,

have a higher heat exchange weight, and have less fouling are more appealing in this regard.

1.2. Heat Exchanger

“A heat exchanger is a device that transfers heat from one medium to another. A strong separator might be used to keep the media from merging or coming into direct touch. Space heating, cooling, testing, material plants, petrochemical plants, oil refineries, oil gas, cryogenics, and water treatment are all typical applications. The usual instance of a heat swap is the thermal exchangers in a car that interchange heat and air via the furnace and the hot engine cooling fluid”.

1.2.1. Flow Arrangement

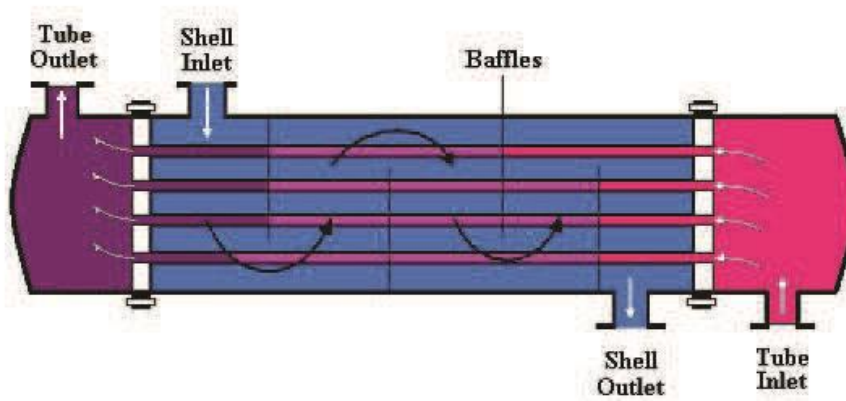


Figure 1.1: A Shell and Tube heat exchanger

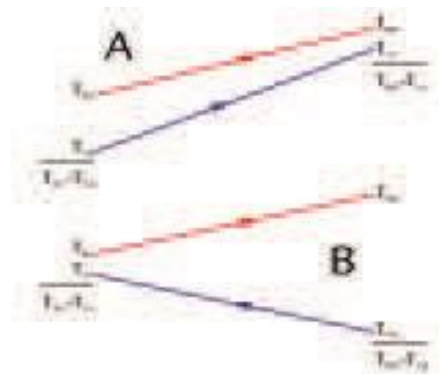


Fig 1.2 Countercurrent (A) and parallel (B) Flow

Depending on how the heat exchanger works, there are two basic categories. There are situations when a parallel flow exchanger is appropriate. The exchanger can be employed when both liquids flow in the same direction. Fluids enter the exchanger through a different set of valves in counter stream heat exchangers. Exchange the most heat to extract as much heat as possible from the heat medium. Counter-trading: a counterflow strategy In the cross-section of a heat exchanger, liquids are frequently changed. Warm exchangers operate because they wish to elevate the surface that separates the two liquids, making it easier for fluid to flow through them. It may also be difficult for the exchanger to function effectively if one or two coils of blades are enlarged, increasing the surface area and perhaps directing the liquid stream or causing disturbances in the liquid.

As a result, it is possible to describe the driving temperature on the heat exchange surface as an adequate method of fitness. When it comes to basic frameworks, this is the "upper temperature contrast log" (LMTD). The NTU Strategy is utilised over and over again because LMTD is inaccessible.

1.2.2. Types of Heat Exchanger

- Heat exchanger with shell and tube
- Heat exchangers with plates
- Heat exchanger with plates and shells
- Heat exchanger using a diabatic wheel
- Heat exchanger with plate fins
- Heating exchanger with pillow plates
- Heat exchangers for fluids
- Warmth-restoration devices for waste
- Heat exchanger with a dynamic scraped floor
- Heat exchangers that work on a phase-exchange basis
- Heat exchangers with direct contact
- Heat exchangers with micro channels

1.2.3. Shell and Tube Heat Exchanger

“The heat exchanger's shell and tube include a number of tubes. A set of these tubes contains the heated or cooled fluid. The second fluid heats or absorbs heat by flowing through the heated or cooled tubes. A tube bundle is a collection of pipes that might include a variety of pipe types. High pressure (pressures exceeding 30 bar and temperatures over 260 ° C) is often used in shell and tube heat exchangers. The heat exchangers have a sturdy tube and shape”.

Many thermal structures should be addressed when developing heat exchangers. Through perforations in tube sheets connected to water containers, the latter section of each tube may be seen. The tubes might be different depending on the need.

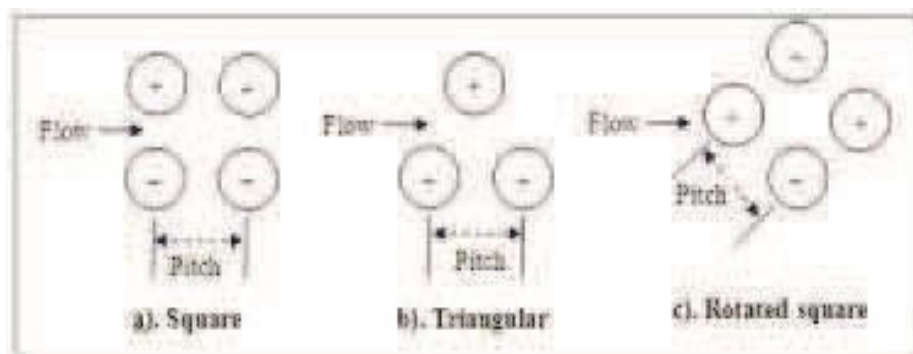
- **Tube diameter:** Heat exchangers that are both inexpensive and compact are those that have a small cylinder diameter and a narrow breadth. However, it fouls up more quickly, and since it is so little, it is more difficult to clear mechanical fouling than other types. It may be essential to increase the diameter of the cylinders in order to reduce clogging. As a result, while calculating the cylinder distance, it is necessary to take into consideration a variety of different factors.



“Figure 1.3: Tube bundle and insertion”

- **Tube thickness:** The thickness of the wing may be estimated using the following factors: -

- Tubes corrosion of steel
 - Vibration generated by flow had resistance.
 - Tube axial strength
 - Components are readily available
 - Strength of the hoop
 - resilience under pressure
- **Tube length:** Smaller-diameter heat exchanger shells will be less expensive. Heat exchangers are built to endure as long as possible as a consequence. The manufacturing capacity, on the other hand, has not been surpassed. If there isn't enough room on the site, it may be necessary to use pipes that are twice if necessary. Large, thin tubes are often difficult to remove and reinstall.
 - **Tube pitch:** “When creating the cylinders, the distance between the cylinders' centers should not be less than 1.25 times the cylinders' outside areas. A larger tube pitch results in a larger overall shell width, making the heat exchanger more expensive and bulkier”.
 - **Tube corrugation:** This kind of cylinder aids in the heat exchange process by increasing the turbulence of the working fluids, resulting in better results.
 - **Tube Layout:** Triangular (30°), turned triangular (60°), square (90°), and pivoted square (45°) are the four basic cylinder designs. The heat exchange in the triangle instances is more noticeable because the liquid is forced to flow in a more turbulent manner around the funneling. Cleaning is more common in square instances, which have a significant fouling influence.



“Figure 1.4: C Types of pitch layout”

1.2.4. Plate Heat Exchanger

Another sort of heat exchanger is a plate that resembles a heat exchanger, such as this one. One is made up of many small, slightly insulated plates with large outer regions

and holes for heat to pass through. This may be more efficient than the heat exchanger on the shell and rotor plate in some situations, but it may not be. When using gaskets in open circles, they must be removed, cleaned, and checked on a regular basis. This is known as "plate and frame" work, and it is commonly found in the HVAC work. It's also known as a plate and a structure, but they're not the same thing. Brazed plungers and vacuum-brazed plate arrays are two examples of their many shapes and sizes. These heat exchangers can be used for a variety of purposes, including cooling. Plate heat exchangers differ based on the type of plate used and how the plates are arranged. Some plates may have chevrons, while others may have sheets or grooves.

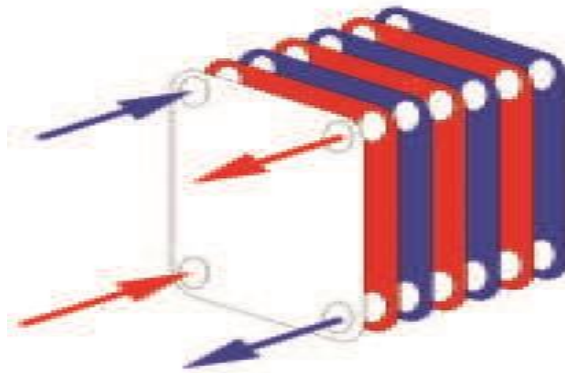


Figure 1.5: Plates of heat exchanger.



Figure 1.6: single plate of heat exchanger

1.2.5. Waste Heat Recovery Units

Waste heat recovery parts are heat exchangers that employ water or oil to remove energy from an active gas stream. It is also feasible to generate heat from waste gas utilized by gas turbines, diesel engines, and refineries.

1.2.6. Dynamic Scraped Surface Heat Exchanger

The "(dynamic) scratched heat switch" is another kind of heat switch. When it comes to heating or cooling, it's often used with materials that crystallize, dissipate, or get severely fouled. There are long-lasting events because the surface is scratched consistently, deflection is avoided, and a suitable heat exchange rate is achieved. The equation developed for this will be

$$Q=A*U*LMTD, \quad 1.1$$

❖ Where Q= heat transfer rate

1.2.7. Phase-Change Heat Exchangers

Using thermal swaps, you may heat a liquid, a vapour may be vaporized or refreshed while also being collected. Reboilers are used in material factories and refineries to provide steam to refining towers.

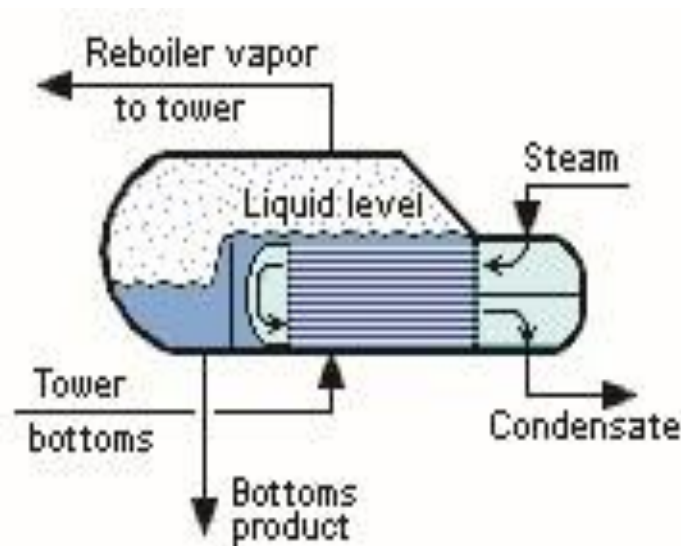


Figure 1.7: The reboiler kettle in an industrial distillation tower

distillation vapour is re-solidified by condensers in refining systems. Heat exchangers in steam are often used in power plants with steam turbines to bubble water. Hot exchangers or related machines for producing water steam are often referred to as boilers or steam generators.

There are separate large heat exchangers called steam generators that move heat from the main frameworks of nuclear power plants (such as the reactor plant) to the secondary frameworks (such as the pressurizing water reactors) of the plants (steam power plant). Both nuclear and fossil fuel turbine condensers use surface condensers to turn steam into water that can be used again.

Regenerative heat exchangers can be used to swap heat between streams in distillation cooling and warmed reboilers, for order, in order to refresh energy and reduce cooling expenses. They can also be utilized for other purposes, such as lowering energy expenditures in cooking and other plants.

1.2.8. Direct Contact Heat Exchangers

The hot and cold streams come into direct contact without insulating separators during the second stage of a heat exchanger. This results a heat exchange. As a result, these energy exchangers may be better understood:

- “Gas – liquid
- Immiscible liquid – liquid
- Solid-liquid or solid – gas”

The Gas-Liquid classification is often used in direct contact heat exchangers to exchange heat as a drop, film, or shower between a gas and a fluid. These heat exchangers are used to chill, moisten, cool water, and nourish plants with nutrients.

Direct heaters aren't the same as "rods." Instead, they're gas-to-air heat exchangers, which are often made of stone-tolerant pads. In several living rooms, heat exchangers go through the mill. On one side of the heat exchanger, air passes through the ignition components, while on the other side, air passes through the heat exchanger. As a result, a split heat exchange is a potentially hazardous circumstance that must be addressed since burning things enter the structure rapidly.

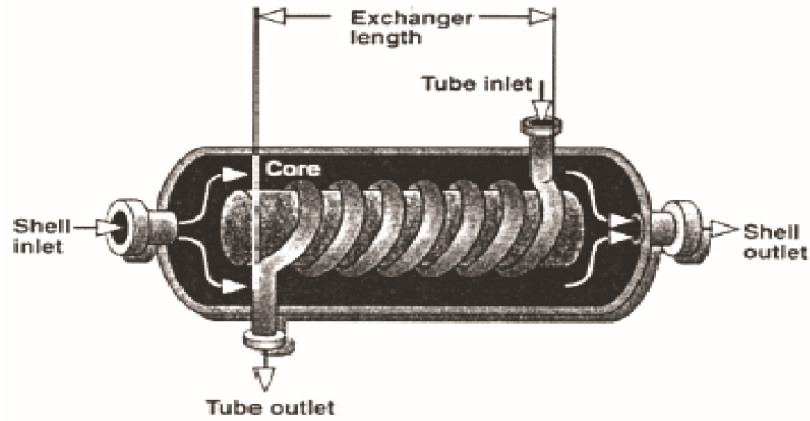
1.2.9. Helical Pipe Heat Exchanger

Maintenance control, nuclear reactors, conservative heat exchangers and heat recovery systems are only few of the areas where helically bowed tubes are employed. Bended cylinders are required for the therapeutic network since fluid streams are present in various mixing tubes. Heat exchangers, for example, find helical curls especially attractive.

Because helical tubes are widely used in these applications, knowledge of the weight drop, stream flow, and heat exchange properties is essential. Weight drop qualities are necessary for evaluating syphon control, which is required to overcome weight drops in order to provide the appropriate stream rates. The cylinder's form is also being manipulated by these weight reductions. The ebb and flow inspires alternative stream patterns that run in the opposite direction of the main hub stream. Typically, liquid in the cylinder's core flows towards the exterior divider, then flows back to the cylinder's internal portion by streaming back along the divider, as shown in Figure 1.8.



“Figure 1.8: Helical coil type pipe”



“Figure 1.9: Helical Pipe Heat Exchanger”

The baffle is vital in this design because it prevents the liquid from passing directly through the shell and contacting the coil only briefly. Baffle: This one alters the rate at which liquid travels around a loop. There will be zones where the water does not move between the bends. Coil efficiency would be reduced in these places because heat would have to flow through the liquid, lowering coil efficiency. Another method for calculating the exterior heat exchange coefficient is to follow a stream of water through a bank of non-stunned round cylinders. This is yet another method for accounting for the complex geometry. Consequently, a major downside of this kind of heat exchanger is the difficulty in predicting thermal exchanges and the available surface area for heat exchange. With little data on calorific exchangers, as well as with irregular flow throughout its perimeter, these challenges are caused.

CHAPTER 2
LITERATURE REVIEW

CHAPTER 2

LITERATURE REVIEW

2.1. Literature Review

This section summarizes the results of two measurements and three measurement displays related to heat exchangers and Computational Fluid Dynamics (CFDs) in shell and tube. In order to have a deeper understanding of the subject matter being researched, it is also required to consult related studies.

M.D. Rajkamal et. al. (2018) This shell and tube heat exchanger research and programming employed programming for a helical coil heat exchanger with 30o edges. Carbon steel is now widely utilized in place of copper in a variety of applications. Both of these materials perform better in terms of heat exchange and are more resistant to erosion than copper. A laminar stream of liquid is used to heat the space. Heat exchange rates increase when the mass stream rate increases, which is also being investigated.

Shuvam Mohanty et. al. (2018) Counter stream heat exchangers in cylinder warm exchangers with rotating streams are studied numerically here in this work. The study looks at both heat exchange and weight loss in a heat exchanger used to evaluate water as a liquid. According to a number of specialists, counter-stream warm exchangers may be improved if the liquid temperature was adjusted and various liquids were employed. Because of the wide range of temperature and mass stream rates, it was utilized to forecast how the heat exchanger will perform in terms of temperature change and weight loss. This was accomplished using CFD software. Using CFD simulation, the two-pipe counter-stream heat exchanger's effects are established after approval of the CFD investigation's philosophy. The turbulence model section is a good fit for these characteristics and consequences.

Ankush S. Patil et. al. (2017) There are a variety of heat exchangers that may be used to transfer heat from one medium to another without mingling them. Either removing heat from or adding heat to a liquid might be the goal. Further improvements to heat exchangers are being investigation to the point of absurdity. It has been explained in this inquiry how various professionals have examined the adequacy of the heat

exchanger. Folded plate heat exchangers have been shown to have the highest rate of heat exchange.

Chandan Kumar Sethi (2017) In this experiment, the working fluids are saltwater and motor oil. A plate heat exchanger is employed in this experiment. The convective heat exchange coefficient, the general heat exchange coefficient, the exchanger adequacy, and the liquid yield temperatures, as well as the liquid temperatures, are all examined in this examination. Changes in parameters such as flow rate and temperature of hot and cold liquids are used to execute tests. The study was carried out using an ANSYS 12 CFD model. The data is utilized to calculate various parameters, and the relationships between different values are displayed. Plate Heat Exchangers have been extensively researched and argued in order to determine the optimum technique to employ them.

Deepa Shrivastava et. al. (2017) Computational Fluid Dynamics has made it possible to anticipate how heat exchangers will flow and how successfully they will transfer heat to and from other properties. ANSYS is used to create a three-dimensional matrix for an in-tube heat exchanger to improve its performance. ANSYS is used to understand how objects move, how powerful they are, and how much vitality they consume. To validate CFD, testing is required for both the parallel and counterstreams. Results from the CFD simulation of heat transfer were found to be very close to those obtained from experimental. Tube-in-Tube heat exchanger CFD reconstruction done using ANSYS programming for Tube-in-Tube heat exchanger. CFD results for heat exchange properties are compared to experimental findings (Nusselt number). The results are presented in a graphical format, and they appear to be in remarkable agreement.

Dipankar De et. al. (2017) “ANSYS FLUENT programs with a helical confounding, comparing, and plainly shocked by CFD tests are used to investigate the construction of shell and cylinder heat exchangers”. There are seven copper tubes with a 20 mm outside diameter, an inner width of 17 mm, an internal coat of steel width of 600 mm, an inner steel shell width of 90 mm, and an exterior range of 110 mm in this design. Models are set up using CATIA programming tools. The helix point where the stream intensity remains constant results in a weight loss and a change in the heat exchange coefficient.

K. Venumadhav et. al. (2017) In order to get the highest possible efficiency, a Double Helical Circular Pipe Heat Exchanger was designed and disassembled. Additional items have been relocated and deconstructed in this layout. Steel, aluminum, and copper are the primary materials. SolidWorks software was used to complete the design of the Double Helical Circular Pipe Heat Exchanger. ANSYS software is used to perform the test at this step. If you want to get the most out of your materials, utilize a counterflow heat exchanger rather than a parallel flow one. Because copper is more durable than aluminum or steel, it is the best material.

Mohammed Irshad et. al. (2017) The primary purpose of this research is to construct a heat exchanger for shells and ropes. It will design various assemblies using the ANSYS programming tool, taking into consideration how flow moves via pipes and tubes as well as how hot and cold they are. There are five ways to do research at the organization. The critical geometry of the shell is shown and crossed when using the CFD kit ANSY S14.5 to rebuild the shell and cylinder warm exchanger. Each of the heat exchanger's seven tubes is 600 mm long. The heat exchanger shell, on the other hand, is 90 mm long. We demonstrate how many factors can affect the temperature, weight, and speed of the shell while it is being manufactured.

Q. Miao et. al. (2017) In this research, investigate into why an air heat exchanger failed. It's used to make a three-dimensional finite element model of an air heat exchanger's natural frequency and vibration mode. To investigate a flow-induced vibration, the results of a modal simulation are used. The multi-physical field coupling approach is used to simulate thermal-structural interactions in an air heat exchanger. The temperature field simulation data is utilized to calculate the tension and stress of the air heat exchangers. To determine how long it will survive, a Manson-Coffin model with an identical plastic strain range and cycle life is utilized. A thermal-structural interaction simulation provided us with an analogous plastic strain range. According to the study's findings, thermal stress and vibration generated by air flow are the primary causes of air heat exchange tube fatigue failure.

Raju et. al. (2017) In this study, a heat exchanger shell and tube with collaboration devices such as water and steam are used. This is a simplified version of the model. A vapor at a temperature of 140 ° C was used to build a heat exchanger from 40 ° C to 70 ° C in this study. Kern's technique is used to complete the plan, which includes shells,

tubes, perplexities, and other metrics. A PC example was created using CATIA V5 with inferred heat exchanger components.

Samuel Raj et. al. (2017) We will look at the CFD reconstruction of the shell and cylinder heat exchanger and how it works in this exchange. To examine how the heat exchanger works, we use a straight cylinder and a helical one. Straight cylinder heat exchangers (STHE) have a large surface area for heat exchange. They are employed in enterprises because they can efficiently exchange a large amount of heat. The Helical Coil Heat Exchanger (HCHE) is also employed in many contemporary applications because it can fit a greater heat transfer zone in a smaller area and has better heat exchange coefficients, allowing it to be used more frequently. STHE and HCHE demonstrations are compared to demonstrate that HCHE is more beneficial, and the study was completed using ANSYS Fluent 14.5 programming. CATIA V5 software was used to create the model.

Katarki et. al. (2017) Internal and exterior shells of 330 and 350 millimeters were used to simulate the heat exchanger. A total of 36 tubes may be found in the cylinder's 25.4-mm-long barrel. Water and steam are used to finish the shell and the cylinders. Skills may be improved by modifying the structure in light of the findings. Display $k-s$ (RKE) For the best presentation, the best results will come out. Between 356oK and 359oK, the temperature of the cold liquid has been found to be true. 404-403 is fine with hot cash. People say that the hot liquid exchange rate is 11475.76 kW. The fresh liquid exchange rate is 64221.52 kW. The budget has been worked out. Exploration incentive temperature and CFD value are linked in a real way.

B. S. Hasu et. al. (2016) A coating heat exchanger with an internal diameter of UGNX8.0, a diameter of 600 millimeters, a length of 1000 millimeters, and an outside diameter of 12.5 millimeters will be examined, as will a tube heat exchanger with a length of 1000 millimeters and a coat material of steel 1008. By applying the demonstrated procedure, the Shell and Tube with water is finished. In order to do a thermal study of the Shell and Tube heat exchangers, ANSYS programming is used to adjust the configuration. The ANSYS has a relationship to the test outcomes. Using suitable numerical performance, the Shell and Tube heat exchangers program may be improved for increased competence. As a medium with critical current, the types of

exchangers described here have the optimal dimensions and appropriate spreads in cylinder pitch. Analysis's relationship to its format and resources.

Ashok reddy (2016) In this demonstration, CATIA V5 programming devices were used to demonstrate the heat and pressure driven examination of the shell and tube heat exchanger in general thermal coefficients, as well as the nonlinear arrangement of integro-fractional differential logarithmic conditions, which were achieved through the use of the ANSYS programming devices. However, despite the orderly structure, Kern and Bell-Delaware were connected in the structural model that had been authorized.

BalaBhaskara Rao et. al. (2016) A lot of experiments and computer simulations are done to show how varied heater exchangers with different cylinder shapes, like roundabout and circular cylinders, do their job. At the 600 cylinder launch and a 25% cut-off, hot and cold liquids were put to the test. It's possible to figure out heat exchange rates and weight losses for a wide range of situations with Reynolds numbers from 4000 to 20000. Ansys is a numerical testing software. Geometries of elliptical and circular cylinders with 450, 600, and 900 introductions are employed for numerical investigations. As a result of the 25% reduction in uncertainty, startling reductions in action are used. Ansys' calculations are contrasted to exploratory estimations of heat and weight exchange rates on both sides of both shells and cylinders.

VIJAY et. al. (2016) This study uses Genius E., hyperworks, and Ansys software to manage the construction and studies of the thermal exchanger (oil refrigerator) in order to meet the demands of the data gathered from the source and the plans of TEMA and ASME recommendations. The primary objective is to use Ansys to measure the temperature distribution and an opportunity. According to TEMA and the HTRI programming bundle, the heat and pressure drop calculations in this project are done using the precise equation (USA). Weighing 0.29 percent less than the shell and cylinder, it decreases estimations on the side of the shell and on the side of the cylinder. HTRI programming is divided into two categories: 1.4% and 1.68 percent. There is less LMTD and surface area variability with lower temperatures in the separate water bays because of an increase in the oil fouling factor. There is less variation in general thermal exchange. Oil is making a difference in the heat exchange coefficient of a heat stack and in general.

Digvendrasingh et. al. (2016) The primary goal of this investigation is to regulate the heat exchanger utilizing commercial CFD programming, organized using the core approach. It is possible to see the heat exchanger's new shape in symmetric form using Solidworks programming. The temperature and speed profile of the cylinders and the shell are studied using CFD reconstitution in the current study. The rate of heat transfer throughout the cylinder's length varies. It's uncovered, finally. Computer fluid dynamics (CFD) is organized and mimed in the exam shell and cylindrical heat exchanger. A heat exchanger header decision was predicated on a CFD simulation, even though the CFD simulation was used.

Mayank bhola et. al. (2015) ANSYS FLUENT steel program with incorporated air was used to simulate the concentration-tube heat exchanger and the concentrate-cylinder heat exchanger. The air's Reynolds number increased from 21,000 to 100,000. Separately, Gulf-temperature heated water and cooled air at 60 and 26 psi C are available. For parallel and counterflow streams, the analysis includes Nusselt number, heat exchange coefficient, and grating element guarantees.

Pankaj D. Lad et. al. (2015) Condensed heat exchanger rationalization and FEA were discussed in the research. When funnels are used, it is easy to understand how the concentrated heat exchanger pipe operates differently than ordinary pipes. A concentric heat exchanger made of heat pipe heat pipes was used in this study to show how the ASME Code has made internal pipes, tube linings, and fillers thicker to lessen the stress on concentrated heat exchangers. SA 516 Grade 70 was the material utilized. Heat exchanger conduct is examined at various temperatures, with ANSYS results being compared with approved findings. When it comes to ANSYS and judicial decisions, there is considerable unanimity.

Roshan. V. Marode et. al. (2015) It is necessary to develop and produce a test model in order to measure the performance of different structures. Various structures containing two liquids have been thermally analyzed, and a decision has been taken based on the results. This research shows how to prepare and conduct a heated inspection of cylinders. It cautiously follows the same strategy and examines the outcomes. The ANSYS programming breaks down when it comes into touch with the identical design tubes, and the results are reflected.

Saurabh Kumar et. al. (2015) In this case, the two-cylinder helical coil warming exchanger was modeled using the ANSYS FLUENT 15 computer program when it was exposed to a shielded external splitter. The graphs depict the Nusselt number and the heat exchange coefficient under turbulent settings. To determine which management demands were most critical, the typical violent $k-\epsilon$ model was integrated with the finite volume technique. The results reveal that the Nusselt Nanofluid number is larger than the base fluid nonfluid number, with SiO₂ nanofluid generating the highest heat exchange improvement for CuO and Al₂O₃. The Dean Number and volumetric concentration may be expanded to improve heat exchange.

Sunil B. Revagade et. al. (2015) Plan and thermal study of several cylinders are shown here. For a limited time, the same processes and results are being evaluated. Using ANSYS programming to identify tubes of the same construction and comparing the results. Because of this, we infer that the deformed cylindrical cylinders have higher heat exchange values than others, i.e. 1.14 more than others. For water, they look at the data and see what we can learn from it (Case-I). In terms of viability, the curve of the cylinder is 1.17. So, based on my findings, I recommend using a wrapped cylinder. As a result, it's important to have a thorough understanding of the exchanger setup guidelines to get the most of this product.

T. M Shaikh et. al. (2015) To investigate how heat travels through the helical coil heat exchanger, a computer simulation was employed. The results of the CFD investigation are thought to be in good agreement with the experimental findings. Exorbitant inquiry and lengthy calculation may be replaced by CFD Analysis. Experiments and CFD simulations have shown that growing mass is correlated.

Jayachandriah et. al. (2014) The shell is made of Steel 1008 and Copper-Brass. The internal diameter of the shell is 400 mm, while the length of the shell is 700 mm. The exterior tubes are separated by 12.5 mm, and the tube length is 800 mm. Here's how it works: To test the configuration of shell and cylinder heat exchangers, CATIA V5 is employed. Here's how it works: To show off the finished product, the shell and tube are finished with water. In a heated climate, a computer program called ANSYS is used to examine the shell and tube of a heat exchanger. The test results are compared to one

another using ANSYS. The Shell and Tube layout could be altered to boost efficiency using publicly available data.

Dubey et. al. (2014) The study investigates how various types of loading affect the functioning of the heat exchanger in a variety of ways. The employment of core technology to chill water from 55°C to 45°C using room temperature water was deemed to meet the requirement for a simple heat exchanger shell and cylinder design. We used ANSYS 14.0 to perform a permanent natural thermal test to validate that the structure was correct. In order to create an Equivalent down to earth model, the proper measurement segments were used. An aluminum foil heat exchanger was tested in a variety of streaming situations utilizing different materials such as wool cotton and bandages. They also looked at the heat exchanger's performance at a variety of ambient temperatures to determine how it was affected. Additional turbulence and its effect on pump adequacy were also studied by shutting down the pump's opening. Detailed discussions of all of these perspectives and the speakers who expressed them took place throughout the research.

2.2. Conclusion

Heat Exchangers system literature is used to determine how well a building's Heat Exchangers system performs in practice. The varied forms of supply water stream containers employed are linked to the findings of the cooling system's performance and thermal comfort question. Heat exchanger tubes were discovered to be available from a variety of sources. The heat exchanger design and analysis software ANSYS CFD is a good example of this. In a variety of heat exchangers, three equally sized chambers are employed. Heat exchanger room simulations are carried out using ANSYS CFD.

CHAPTER 3
FINITE ELEMENT METHOD

CHAPTER 3

FINITE ELEMENT ANALYSIS

3.1. Introduction to FEM

The FEM is a valuable tool for determining how to tackle a wide range of construction challenges using arithmetic. This approach can handle any complicated shape or geometry for any material, even if there are varied constraints and stacking circumstances at work. The strategy's all-encompassing declaration addresses the need to consider today's complicated construction frameworks and structures, where closed-form arrangements for balancing situations are uncommon. Furthermore, designers must employ parametric structural principles and break down the ideal structure into sections, taking into approach a variety of structural examples (divergent types, materials, charges etc.).

The technique was developed to investigate stress in a complex aircraft structure used in the aerospace sector. It is made using the matrix check approach, which is employed in the production of aviation equipment. Both practitioners and experts have taken notice of the technology. A strategy for a tiny component's basic notion is that a body or system may be divided into small components using certain metrics known as "finite components." With a restricted number of nodes or hubes, the initial body or structure is regarded in a succession of these components.

3.2. General procedure of finite element method

The finite element technique is a piece-by-piece estimating approach that employs a restricted number of measurements for each component. Nodal focus or hub is the term used to describe a grouping of joints inside a structure that are related with a small number of such constrained component parts. A component's base capacity may be used to estimate the range of possible field variables, such as displacement, stress, temperature, pressure, and speed. Initial models are used to assess the effect of hubs on the surrounding environment. System conditions are used to estimate the nodal field variable estimates, which are based on the evaluation of the field conditions.

When the nodal values are known, the approximation functions define the field variable for all element assemblies.

General continuity problems are arranged utilizing finite components technology in a precise and well-ordered manner. Structural solutions may be summarized as follows:

Step 1: - Documentation of the project's steps (domain). As the initial phase in the process of the finite element, it is necessary to break down the result area structure into smaller components.

Step 2: - Selected for the right interpolation method. An acceptable estimate of an uncertain solution can be made since we can't anticipate dislocation (the field variable) in the problematic structure under any particular loading circumstances. The anticipated outcome must be straightforward, and a number of convergence criteria must be satisfied.

Step 3: - The beginning of the part's stiffness matrix and load vector (feature matrices). Displacement should be taken into account when calculating e's stiffness and load vectors $[K(e)]$ and $[P(e)]$. An sufficient allowance for variations should be used.

Step 4: - Assembly of the balance element equations.

Because the structure is made up of multiple end components, stiffness matrices and load vectors are used to design the individual pieces, and the overall balancing equation is stated as,

$$[K]\phi = P \dots \dots \dots (3.1)$$

ϕ is referred to as the nodal vector, and P is the full-form vector or nodal pressure when $[K]$ is referred to as the assembled rigidity matrix..

Step 5: - Calculation of the nodal displacement values by using a system equation (subject variable). Boundary circumstances must be taken into account in the basic balancing equations. After the boundary conditions have been included, the equilibrium equations may be stated.

You may easily solve the "" function for linear equations. Non-linear issues, on the other hand, may need a phased approach in which the stiffness matrix [K] and/or weight vector P are adjusted one at a time.

Step 6: - In order to calculate the stresses and strains of the details, It is possible to derive the fundamental mechanical equations for stable or structural mechanics by taking into account the Nodal motions. The bracketed phrases in the stages above follow the standard FEM step-by-step procedure.

3.3. Convergence requirement

The finite element method provides a numerical solution to a complex issue. When it comes to integrating with the structure, this may be a regular layout. The structure should be accomplished when the work is enhanced and These requirements would be fulfilled if the displacement function was met.

- The displacement characteristic must be continuous. This problem may be readily solved by using polynomials in the displacement version.
- A stiff displacement of the information frame will be represented by the displacement function. This occurs when the nodes are forced together in an inflexible frame movement; the item should not be exposed, and the nodal powers should be null. This is generally ensured by the standard phrases used in displacement mode polynomials.
- The shifting feature should be able to show the unit's continual tension. The objective of the condition may be understood if we consider a circumstance in which the frame or form is divided into smaller and smaller pieces. Because these components have an infinitesimal size technique, the strain in each element is also technically constant. The linear terms in polynoms satisfy the requirements for 1-3-D elasticity.

3.4. Advantages of FEM

The attributes of each constituent are evaluated independently, allowing the material features of each piece to be clearly integrated. Because each FEM numerical estimate is based on the idea of description, arbitrary and irregular shapes no longer pose a challenge. The multi-dimensional constant is seen by innovation as either

diversity or staying approach. As a result, there is no need to extend the approximate answer to every point in another input cycle using a continuum.

One of the key benefits of FEM is the ability to employ limitations in integrated computations. This is a straightforward procedure that does not need the use of advanced technologies. Before they can be tested, algebraic equations of finite elements in each body must meet specific constraints.

3.5. Applications of FEM

Air ship construction tests, such as static wing tests and rocket builds, dynamic examination, reaction to arbitrary weights, and occasional burdens, are among the best methods. The finite element approach has a restricted number of elements. Using a finite element technique, mechanical planners can successfully deal with stretch-focus difficulties, push weight vessel analysis, and dynamic mechanical link testing.

An explicit application of the finite element approach is needed to address three key kinds of limit estimate concerns: harmony of permanence, time-independent issues, worries about self-value, and spread or transitory difficulties. Strong mechanical issues, temperature, or thermal motion transfer might be exacerbated by evident relocation or stress dispersion due to the heat exchange problem. In the strong mechanics or basic problem, the body reaction is linked to the eigen valuation problem. Because there is no liquid mechanics problem, laminar streams are more reliable, and reversing characteristics are discovered because there is no electro circuit problem.

Finally, FEM's utility in structural building is limited to the analysis of built stands, casements, and bridges. The study of dynamic stresses requires a lot of structure. This activity assists you in determining normal stress frequencies, modes, and structural reactions. The design also employs restricted component technique principles in both the static and dynamic presentation of its structures, such as atomic vessels, control structures, and the dynamic response of the reactor segment's regulatory structure to changes in its structure. Furthermore, even the biomedical construction adopts minimum components approach for effect study on skulls. Geomechanics may be linked to the unique strategy of its components via exhumation, sub-ground study, and sophisticated analysis of the dam supply system.

3.6. FEA SOFTWARE – ANSYS

Dr. John Swanson is the creator of ANSYS. Inc., which thought up with the notion of selling PC-simulated building blueprints in 1970, was one of the first companies to employ FEA. Ansys, Inc. assists and provides businesses with adaptable, broad-based design frameworks that assist them in unraveling all types of research difficulties as well as expanding their programming and equipment interests as part of its work. If you work with ANSYS, Inc., you get to decide the pace. You can find it right here. Customers that utilize a procedural approach to structure and assembly might avoid costly and time-consuming "building and interrupting" cycles. Clients benefit from the usability, information similarity, multi-stage assistance, and multilateral scientific combined field capabilities of an ANSYS examination and reproduction equipment.

3.7. Evolution of ANSYS Program

ANSYS has evolved into a multi-use strategy research programme that has taken into account a wide range of strengths from across the globe. Today's software is cutting-edge and simple to use. Each discharge's new and updated features make the software more adaptive, effective, and efficient. ANSYS helps professionals to meet with contemporary weights and needs in various areas.

3.8. Outline of the program

The kit is precise, durable, and has a built-in design examination. On big computers and systems, the software is executed from PCs to workstations and supercomputers. ANSYS provides file consistency across all platforms and software families. Customers may input plan models supported by a PC into ANSYS and do rehashed work because to ANSYS' structural information. As a result, all ANSYS users may expect a broad range of adaptive design responses.

User Interface: Despite its extensive and sophisticated functionality, its graphical user interface makes it simple to understand and operate.

The ANSYS application may be instructed using one of four graphical methods:

- ✓ “Menus
 - ✓ Dialog Boxes
 - ✓ Tool bar
 - ✓ Direct input of Commands”
- **Menus:** An analysis programme or a set of related functions may be found in a single menu.
 - **Dialog boxes:** These are the types of windows that enable users choose whether or not to perform a job or declare their preferences. The customer is asked to commit to a certain capability in these boxes.
 - **Tool bar:** The ANSYS program's toolbar is an incredibly efficient method of executing the program's commands. However, directives are utilized to provide and monitor all system capabilities, notwithstanding their commitment.
 - **Output window:** The ANSYS feature Response may be recorded.
 - **Graphics window:** Show or visually express test results in the space designated for practical displays. In order to accommodate individual preferences, the client may alter or reduce the window's length.
 - **Input window:** Displays programme prompt messages and provides an information space for compiling ANSYS instructions.
 - **Main menu:** The essential ANSYS capabilities, organised on side menus, are included into the program's movement.
 - **Utility menu:** Mapped to an ANSYS session, this includes the ANSYS programming capabilities. These skills are implemented via the use of smooth down-menus that lead to an activity or dialogue.
 - **Processors:** ANSYS power coordinates two meetings with the names of processors. As an example, the ANSYS software has two post-processors and a few helper processors that use a structural streamlining agent. For this configuration, the client may create a tiny component model using the ANSYS preprocessor. The model's response is determined by the processor's utilisation of heaps and limitations.
 - **Database:** A single database is used by ANSYS to keep track of all model data and solution tests. People use a processor database to keep track of the data for the models they make (including solid model and geometry of finite elements, materials etc.). A solution processor is used to write down data about loads and how the solution worked out. A post-processor is a tool that is used to write down data after

it has been processed. As a result, data that was written to the database by one CPU can be read by other CPUs.

- **File format:** Data may be transferred from one portion of the application to another and saved to the database using files as necessary.

3.9. Costs of ANSYS conception and development are reduced (FEA)

PC models or CAD model exchanges can be used to create PC models, run loads, perform other design work, and concentrate biological responses such as fear, temperature transfer, or the influence of attractive vector fields. They can also be used to create computer models. It is nearly impossible to test models in some circumstances. ANSYS has been employed in a variety of biomechanical applications, including the replacement of the intraocular focus. The continuous coal mining machine's gear, integrated circuit board, and bit holding are only a few examples of other uses. In order to reduce the quantity of costly models, adaptability and inflexibility needed to fulfil their objectives, and identify the most effective architectural alterations, ANSYS has developed the ANSYS Plan. Innovative businesses strive to find ways to provide the highest-quality products at the lowest possible cost. Construction and assembly expenses may be reduced by using ANSYS (FEA), and architects can have more confidence in the products they design using ANSYS (FEA). To utilize FEA, you must have the right software. When using the prototype later in the assembly process, it's also a good idea to double-check the final plan.

3.9.1. Program availability

Pentium ANSYS may be used on Windows 95 or NT-based PCs, workstations, and supercomputers that are mostly UNIX-based. ANSYS Inc. is always dealing with new technology and systems.

3.10. Material properties

“For a static analysis, Young's module (EX) must be specified. Density (DENS) and mass attributes such as density (density) are described when we cannot apply delay charges (such as gravity). The thermal extension coefficient (ALPX) is also characterised on the occasion that we propose to apply thermal load (ALPX occurrence) (temperatures)”.

3.11. Obtain the solution

Shape and analytics options are described, then loads are used to begin the finite elements solution:

- ✓ “Pre – processor step
- ✓ Solution step
- ✓ Post-processor step”

3.12. Pre – Processor

For a comparable application, a preprocessor provides fewer PC frames by centralizing and centralizing. This basic and modest model swapping between frames. FE models (which have a restricted number of components) may be easily set up using Preprocessor's user-friendly software. Preprocessor-generated information is used to build the schema when an issue is identified. The temperature and other information is shown on the screen.

- **Geometrical definitions**

Common focuses, axes, territories, and volumes are among the preprocessor's four distinct geometric components. The geometric picture of a building may be achieved using these materials. There are no additional substances in any of the materials.

- **Model generations**

Two separate strategies are used to generate a model:

- a) “Direct generation.
- b) Solid modeling”

You may build up controls over component size and form to educate ANSYS to construct each hub and component in accordance with the model's geometrical limitations using solid modelling. The immediate generation strategy, on the other hand, allows us to determine the size, shape, and network of each component before displaying them in ANSYS. Even Nevertheless, it is possible to imagine a future in which the information age is controlled by computers (by utilising directions, for example, FILL, NGEN, EGEN and so on.) An instant age approach is basically a genuine numerical method that necessitates the tracking of all hub numbers as the work

on the restricted portion is built up. To accurately represent faults in a model, a detailed examination is necessary. For model development, direct age tends to encourage modelling faults because of its lack of innovation and flexibility. Product strategy normally dictates the use of strong displays since they are more fantastic and flexible than traditional displays of the same age.

- **Mesh generation:**

Finite element analysis begins with a focus on a limited number of focal points known as nodes, from which discrete components known as components may be extracted. Load limitations will be assigned to such components and nodes. A mesh framework is a collection of these components.

- **Finite element generation:**

In a final component inspection, the greatest amount of work is spent developing components and nodal information. Customers may use Preprocessor to design hubs and components that provide them control over the size and quantity of components while they wait. Components of different types may be mapped or created on geometrically based elements.

Prior to the restricted component inquiry, the availability, bending file, and other factors should be reviewed to ensure the component characteristics of the components developed with varied programmed component age capabilities of the preprocessor. Instead of describing the nodes separately, programmed work creating preprocessor skills is often employed.

- **Boundary conditions and loading:**

After the part's limited show is over, it must be pushed and stacked with the product. Limits and loads may be described in a variety of ways by the client. All constraints and obligations are considered with the 1D. It aids the consumer in keeping track of load cases.

- **Model display:**

During the design and testing stages of the model, it may be useful to look at it from several perspectives. In reference to the global system, the model must be rotated and interpreted in many ways. The preprocessor is responsible for this functionality.

The window function's preprocessor allows the user to enlarge a specific region of the layout for clarity and detail. Fluidity, scaling, regions, active models, and other capabilities are available in the preprocessor.

- **Material definitions:**

There are nodes in each object that just describe its position. For frame and shell portions, there are no thickness indicators. As an element parameter, its thickness may be set. For a specific collection of 1-D properties, property tables must be created. Various item kinds, for example, have different properties. Beams: sectional cross area, moment of inertia, and so forth. Coatings: Fountains with a high rigidity: none

The customer must also recognize the goods' material features. The elasticity modules and the Poisson ratio are required for linear static analysis. Heat transmission is influenced by the thermal expansion coefficient, densities, and other factors. The elements may have the material attribute set to 1-D.

- **Solution:**

The solution process takes a problem-solving method based on the issue ideas. The computer does all it can to formulate and assemble matrices, and the shifts and stress values are the result. Linear, static, nonlinear, and dynamic analysis are all possible using ANSYS.

- **Post – Processor:**

Interactive colour visuals are used to make the application strong and user-friendly. The findings of the finite analysis may be shown using the tool's extensive drawing capabilities. According to a tabular argument, one of the study's findings (i.e. visual representations of results) shows in seconds what a figurative result takes an engineer to assess. In a jumble of numerical data, the engineer discerns significant components of the testing.

CHAPTER 4

METHODOLOY AND MODELLING

CHAPTER 4

METHODOLOGY AND MODELLING

4.1. Introduction

Computational Fluid Dynamics (CFD) and ANSYS are introduced in this chapter as a foundation for optimising air conditioning analyses. In-depth explanations of the process' various steps are provided. Fluid dynamics and turbulence are involved in the modelling of flow in internal combustion engines. Experimentation and numerical modelling may be used to tackle any physical fluid flow problem. Setting governing equations for each and every fluid mobile, numerical simulation is more conducive to parametric study and provides accurate results. As a result of these developments, complicated flow concerns may now be evaluated and their solutions found in the most efficient manner possible, thanks to advances in numerical approaches. Computational Fluid Dynamics (CFD) has evolved from a purely mathematical field of study to a vital tool in practically every fluid dynamics department. An important role for CFD is to serve as a bridge between experimental and theoretical fluid dynamics. When it comes to waft concerns, researchers had to depend on expensive and time-consuming experiments in the past. The amount of experimentation necessary to understand complex engineering problems has decreased significantly with the introduction of efficient digital computers and numerical simulation methodologies.

4.2. Computational Fluid Dynamics

With computer-aided discrete form solution (CFD), the governing equations regulating fluid flow are simulated in the fluid float hassle. This reduces the amount of time and effort it takes to limit down the design configurations of multiple engineering additives. Using widely available CFD software, Four-stroke DI diesel engine in-cylinder waft system changes were studied using a numerical simulation in three dimensions (three dimensions).

4.3. Basic Structure of A CFD Code

As a CFD coder, it would be difficult and time-consuming to develop one for every novel fluid glide instance. KIVA, AVL FIRE, STAR-CD, Fluent, ANSYS CFX, and Open Foam are some of the typical industrial CFD programmes that have progressed as a result of this. There are GUIs (graphical user interfaces) that are user-friendly and enable the user to employ a built-in code with an array of hassle-fixing features to address difficulties like fluid glide and heat transmission, for example. The three-step procedure of creating a solution to a fluid problem through the usage of a common CFD code is outlined.

1. “Pre-Processor
2. Solver
3. Post-Processor”

4.3.1. Pre-Processor

In the pre-processor stage, the fluid flow issue is evaluated by defining an acceptable geometry. Using a computer-aided design programme, you may complete defining the geometry by recreating an existing model (CAD). A appropriate mesh may be constructed once the geometry has been established. A mesh may be described as a volume or element that has been broken down into smaller ones. As soon as a believable mesh is constructed, further known fluid properties, such as density, viscosity, and turbulent turbulence, are specified. An inlet/exit or a wall-adjacent flux is used to implement boundary conditions such as temperature, strain, velocity profile, and other variables on the cell sides. Preprocessing is the most critical step in developing a solution since the quality of the mesh is so crucial to a realistic solution.

4.3.2 Solver

Solving the transport equations is believed to be the ultimate result of the solver's calculations. The numerical approaches that constitute the basis of the solver are responsible for the following phases in the definition:

- Using basic objectives, calculate the unnamed stream variables
- “Approximations are used to substitute discretization in the governing flow equations and subsequent mathematical operations.
- Solution of the algebraic equations”
 - Numerical solution approaches are divided into three categories:
 - “Finite difference method
 - Finite element technique and
 - Finite volume technique”

Approximating movement variables and discretizing procedures are the main distinctions between the three streams.

4.3.3 Post-Processor

For layout analysis, subordinate factors such as stresses and fluxes should be evaluated. Industrial CFD methods, for the most part, come with their own post-processor, which utilises the nodal records received from simulation to calculate secondary variables and provide a range of visualisations (including contour and line diagrams). That's all of them:

- Domain geometry and grid display
- Vector plots
- Line and shaded contour plots
- 2D and 3D surface plots
- View manipulation and colour postscript output

These features also produce dynamic, animated outcomes. The output of all codes, as well as the visuals, is reliable and may be exported for additional manipulation outside of the code. As a result, all scripts provide record export tools.

4.4. ANSYS CFX (Fluid flow) Workbench Environment

All of ANSYS' engineering simulation software, including ANSYS CFX (fluid flow), is integrated within the ANSYS Workbench platform. With a simple drag-and-drop procedure, users may simply install anything from generic fluid glide assessments of complex interacting systems. Fig. 4.1 shows a fluid waft CFX ANSYS device assignment schematic. It's easy for users to examine the overall performance of several design variables or analyses a variety of different designs. Geometry and meshing tools may be used by programmes from a variety of simulation disciplines in the ANSYS Workbench environment.

4.4.1. Heat Transfer and Radiation: Additionally, ANSYS CFX software has a conjugate heat switch (CHT) feature that may be used to address the thermal conduction in solids, as well. Models for all kinds of radiative warmth alternate between fluids and solids and may be totally or semitransparent to radiation, as well as opaque, can be included.

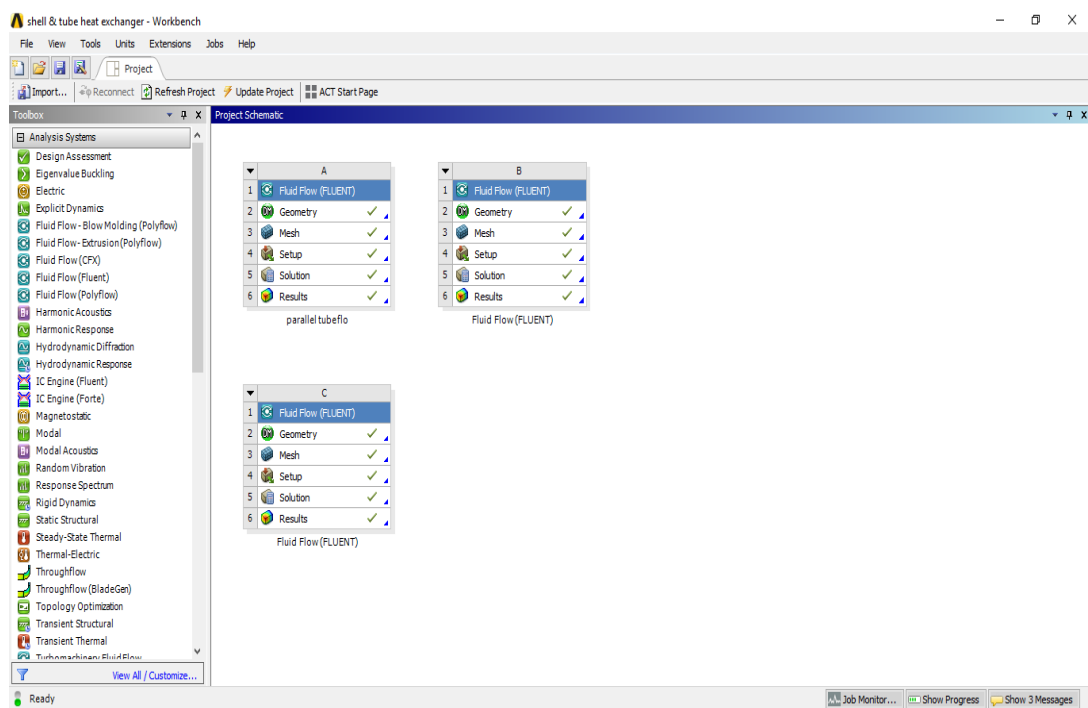


Figure 4.1: ANSYS workbench (CFD)

Project Schematic (Workbench) Fig. 1 shows the fluid analysis tool used in CFD simulation:

- ✓ Make or import a geometric model.
- ✓ The process of creating a mesh for a geometry.
- ✓ Set up the evaluation in such a manner that it may be sent to the solver.
- ✓ To get a solution, control and show the solver.
- ✓ Create a report by visualizing the findings in a post-processor.

4.5. Create a Geometry

“Engineering simulations begin with geometry to represent the layout, whether it is a strong feature of a structural examination or the volume of a fluid or electromagnetic field”. CAD (computer-aided design) systems may provide the engineer with preexisting geometry, or the engineer can start from scratch. For an ANSYS analysis, the Design Modeler in ANSYS opens the door to geometry management in ANSYS. As a result of the need to construct and teach geometry for simulation, ANSYS Design Modeler was developed. The geometry of engineering simulations contains information that is no longer required for simulation. Simulating a fully specified version will increase solver runtimes if just the physics are provided. Doing away with this information might save hours or even days of work in the long run, so it's worth the effort.

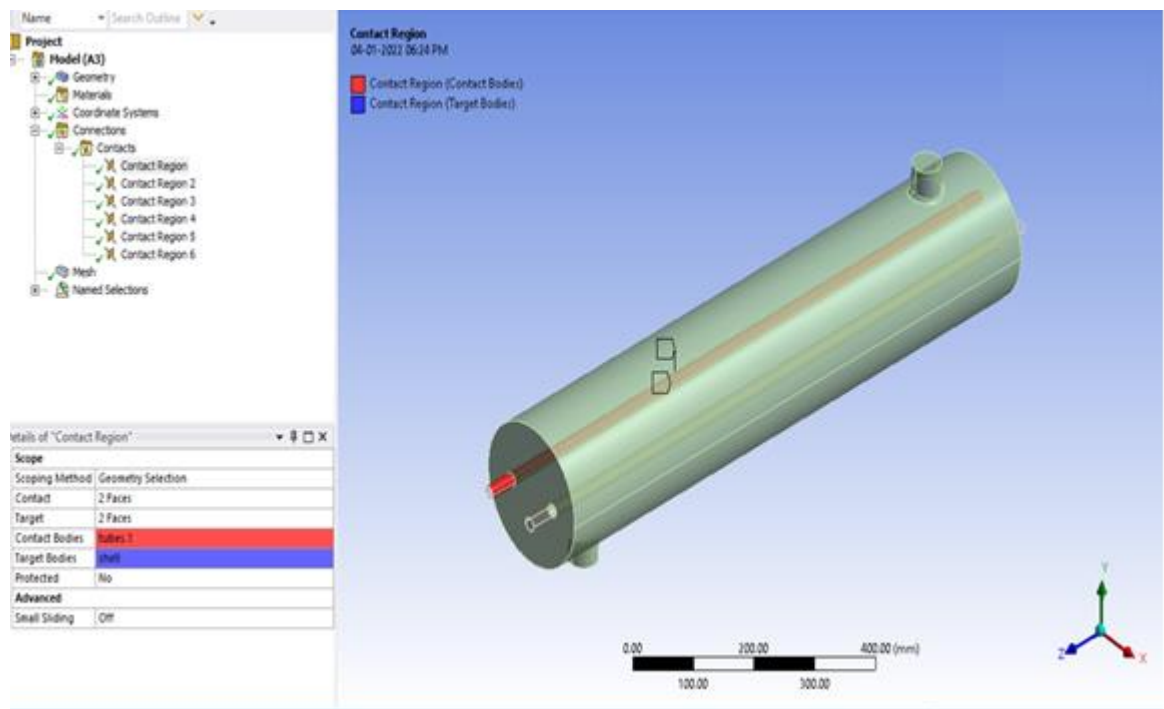
Table 4.1: Dimensions and materials of a shell and tube heat exchanger

Heat exchanger length	1000mm
Shell diameter	210mm
Tube outer diameter	20mm
Tube thickness	2mm
Number of tube (parallel flow)	2
Number of tube (helical flow)	1
Number of tube (pattern flow)	1
Tube material	Aluminum
fluid	Water liquid

As seen in the diagrams shown in Figures 4.2, 4.3, and 4.4, a heat exchanger may be designed with different tube patterns and a variety of functions in the Design modeller and description tree.

➤ **Case 1**

“It is necessary to use parallel flow tubes in this situation. Heat exchangers with parallel tubes are seen in the diagram. In a heat exchanger with an input temperature of 85°C, parallel tubes circulate through the shell. The temperature of the input water is 18°C, and cold water is circulated through the tube as it enters the system”.



“Figure 4.2: Heat Exchanger with Parallel Tubes”

➤ **Case 2**

The second heat exchanger makes use of a flow tube with a 'U' configuration. 'U' pattern tubes are illustrated in the diagram for shell and tube heat exchangers. Passing water through the Shell of Exchanger at 90°C requires the employment of 'U' Pattern type tubes, which are installed in the Shell. Flowing water in the tube has a temperature of 12°C.

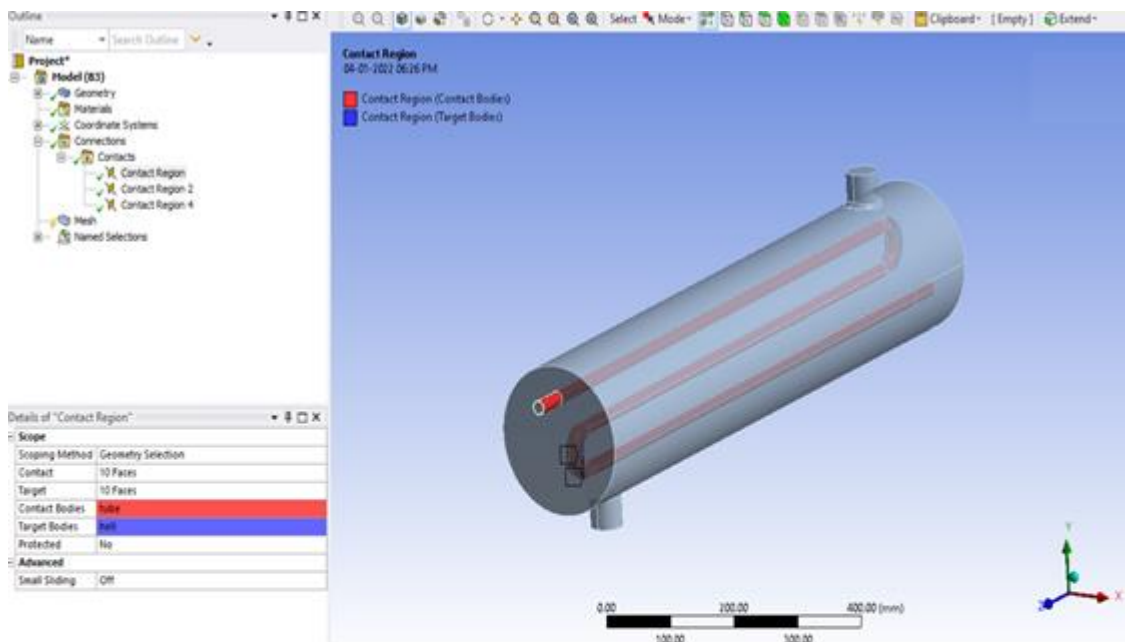
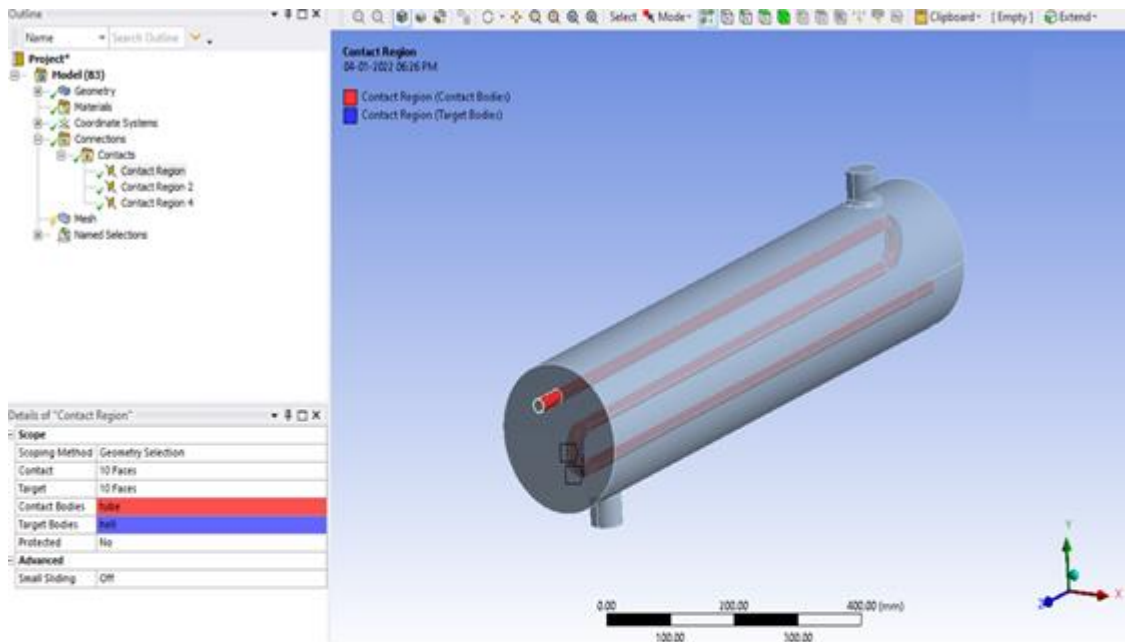


Figure 4.3: Heat Exchanger with 'U' Pattern type flow Tube

The length of the pipe within the shell was increased by connecting two U tubes together in this tube design.

➤ **Case 3**

Helical tube heat exchangers are utilized in this case. The graphic depicts helical tube heat exchangers. The exchanger's intake temperature is 85°C, which causes helical-type tubes to flow through the shell. The intake water temperature is 18°C, and chilly water emerges through the tube.

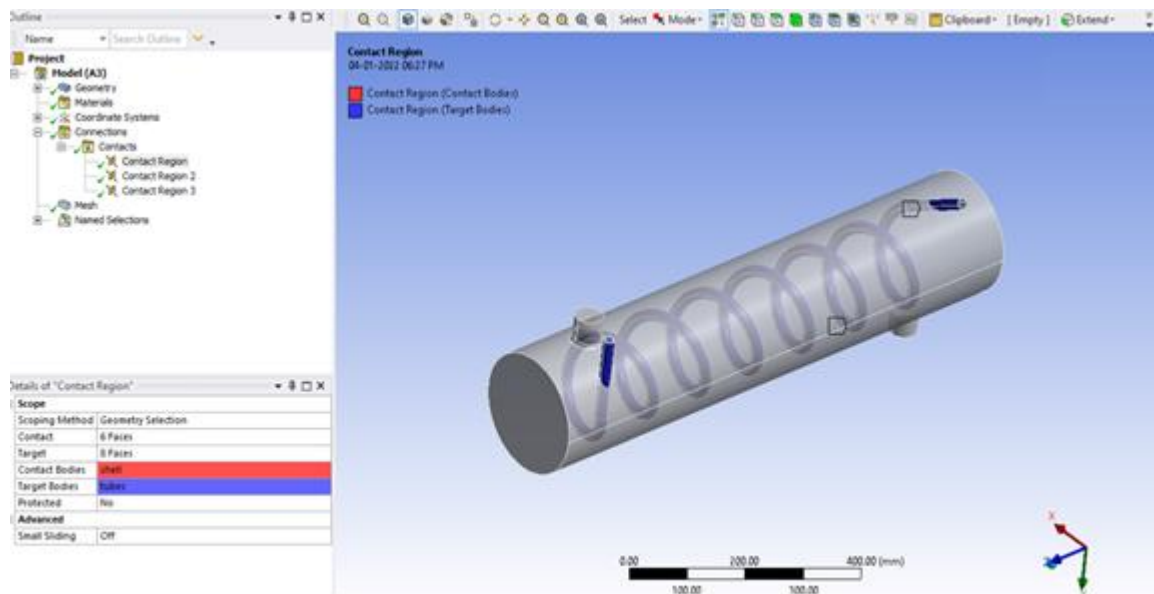


Figure 4.4: Helical Flow Tube Heat Exchanger

➤ **Case 4**

Heat Exchanger No. 4 was designed using zigzag pattern tubes. both the shell and the tube The illustration depicts a heat exchanger using Zigzag design tubes. The water temperature in the exchanger shell is 85°C, and tubes in a zigzag pattern flow through the shell. The temperature of the inlet water is 18°C, and cold water travels through the tube.

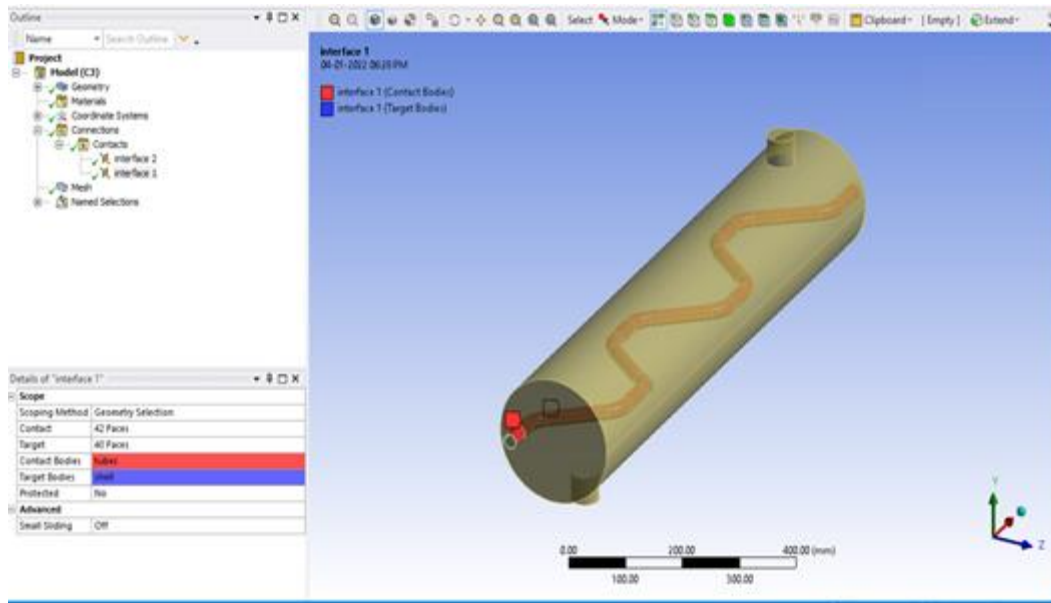


Figure 4.5: Zigzag tube setup in heat exchanger model

4.6. “Meshing of Geometry (discretization of the computational domain)”

The discretization of the computational domain is the most time-consuming and challenging element of CFD simulation. This term is also known as mesh generation. Analytical solutions to the Navier-Stokes equations can only be obtained when the flow is simple. A mesh divides the space into small, finite control volumes using the 'Discretization of the Governing Equations' technique. This method can be used to solve real-world flows. We write down algebraic approximations that can be solved numerically rather than the original equations. When the governing equations are applied to all control volumes, control volume integration occurs, all critical variables (such as mass, momentum, and energy) are discretely maintained.

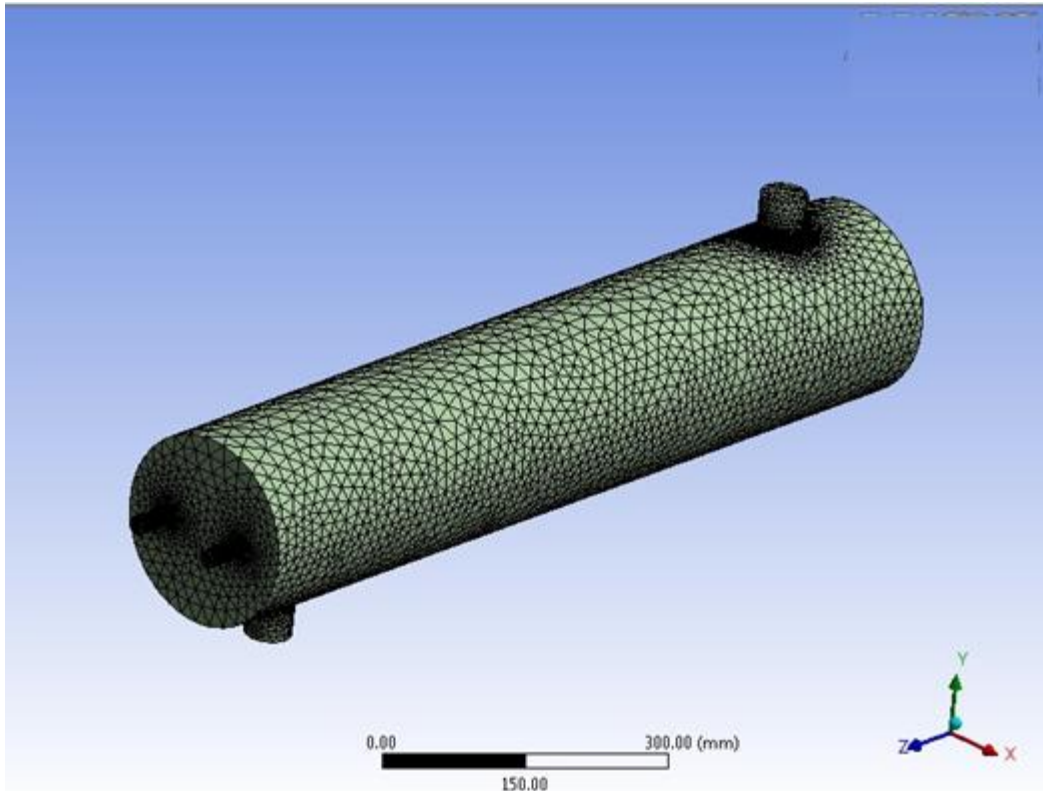
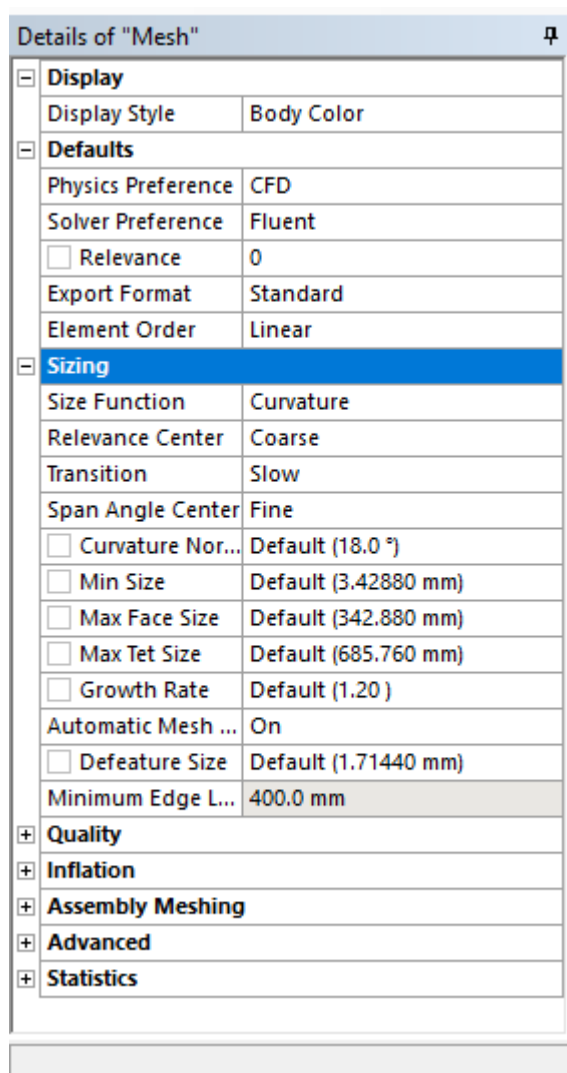


Figure 4.6: Meshed model of heat exchanger

In the finite volume approach, the mesh examines points that create a collection of volumes known as cells. They employ "elements," which are sub-volumes with nodes where the variables are configured. Temperature, pressure, and velocity values will be unique to each element. These values will be displayed in the text. If the mesh isn't good enough, there's a large quality in the CFD output quality.



“Figure 4.7: Details of meshing, CFD simulations of Heat Exchanger model”

In the finite volume approach, the mesh examines points that create a collection of volumes known as cells. The finite element techniques employ elements, which are sub-volumes with nodes where the variables are specified. Each element's temperature, pressure, and velocity will be stated in the dependent variables section. The quality of the CFD output is strongly influenced by the mesh quality. If there are too many cells, the solver will take longer to execute, and if there are too few, the results will be erroneous.

CHAPTER 5

IMPLEMENTATION

CHAPTER 5

IMPLEMENTATION

5.1 Introduction

The different steps of our simulation study on Shell and Tube heat exchangers are detailed in this chapter. For our investigation, we used ANSYS to conduct Shell and Tube heat exchanger room thermal experiments, which is documented here.

5.2. Factors Taken for Result

- “**Temperature:** variations of temperature variations in designed models”.
- “**Density:** It is defined as the ratio of mass to volume per unit of time”.
- “**Viscosity:** Internal friction causes the consistency of a material to become thick, sticky, and semi-fluid as a result of the friction”.
- “**Specific heat:** The amount of heat per unit mass needed to increase the temperature by one degree Celsius is known as "specific heat." As a visual illustration of the relationship between heat and temperature change, the following is often used: The letter c stands for specific heat capacity”.
- “**Thermal conductivity:** the proportion of a material's heat flow rate to its temperature gradient that affects how efficiently it conducts heat”.

5.2.1. Problem Description

Operating conditions of the heat exchanger,

Maximum Fluid Velocity

$$V_{max} = \frac{(U-ST)}{(ST-D)} \dots\dots\dots 5.1$$

“Where U denotes free Stream Velocity (m/s), ST denotes transverse pitch (mm), and D denotes tube diameter (mm)”.

The maximum velocity Reynolds number (Re) is defined as follows,

$$Re = \frac{\rho \cdot V_{max} \cdot D}{\mu} \dots\dots\dots 5.2$$

Where ρ and μ are the density and viscosity of the air, correspondingly.

Using these equations, it is possible to estimate the free stream velocity to meet the Reynolds number range employed in this study. (5500<Re<14500).

5.3. Shell and Tube heat exchanger designing in ANSYS

When creating a product, using computer simulations makes it easier than ever to test and assess how effectively it will keep things warm. ANSYS Fluent was used to perform a CFD simulation of a tube heat exchanger and investigate how different tube layouts affect heat flow. The results reveal that shell and tube heat exchangers with varying tube diameters perform the best.

A copper tube and a steel shell design were chosen from this research, which was then evaluated in Ansys to achieve the highest possible value of temperature changes among the mentioned materials. It's possible that the heat transmission between the shell and the surrounding environment is nil. According to the results of the experimentation, the s-pattern tube design delivers higher heat transmission than other designs in the heat exchanger.

Table 5.1: heat exchanger 3d model dimensions

Heat exchanger length	1000mm
Shell diameter	210mm
Tube outer diameter	20mm
Tube thickness	2mm
Number of tube (parallel flow)	2
Number of tube (helical flow)	1
Number of tube (pattern flow)	1
Tube material	aluminum
fluid	Water liquid

➤ **Case 1**

In the first scenario, a parallel flow tube was utilized to design the heat exchanger. The Shell and Tube Heat Exchanger with Parallel Tubes is shown in the diagram. The entrance temperature of hot water is 85°C for the shell of the exchanger, and parallel tubes flow through the shell. The tube is filled with cold water, and the temperature of the cold water intake is 18°C. Mass flow rate of water taken 0.05 m/sec.

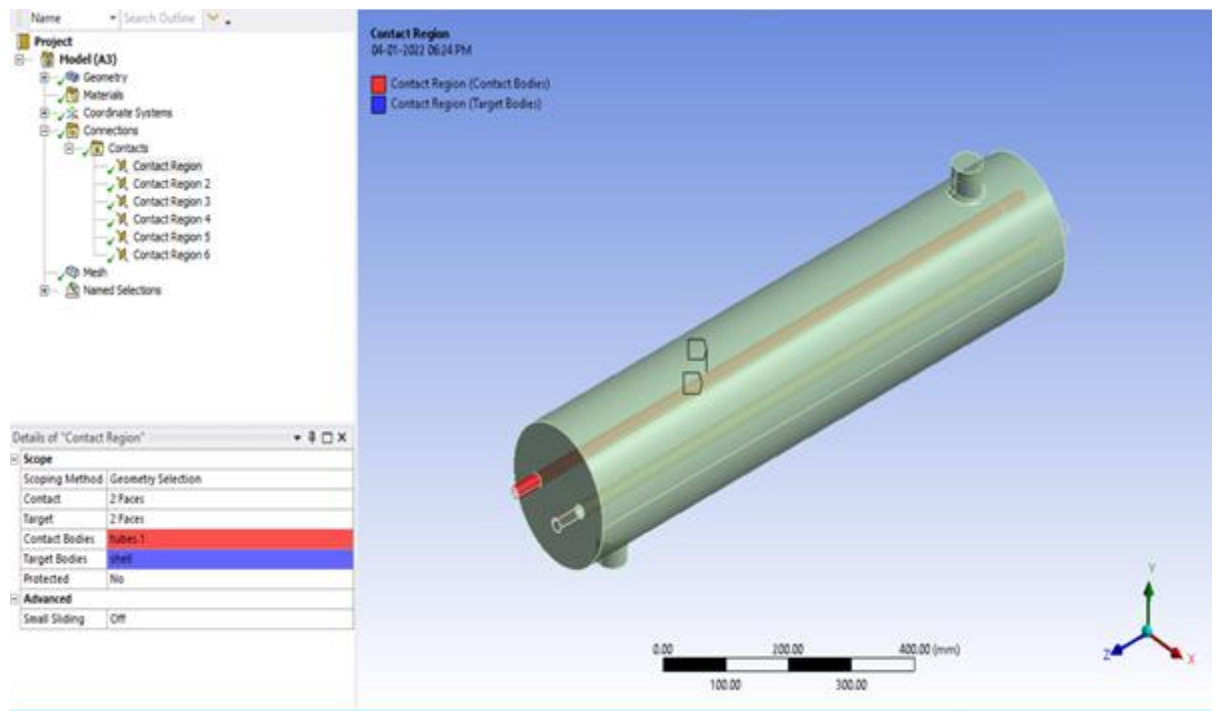


Figure 5.1: Straight tubes used inside the heat exchanger model

➤ **Case 2**

A 'S' Pattern type flow tube is used to construct the heat exchanger in Case 2. The diagram depicts the layout of a Shell and Tube Heat Exchanger with 'S' Pattern type tubes. The entrance temperature of hot water is 90°C for the shell of the exchanger, and 'S' Pattern type tubes run through the shell. The tube is filled with cold water, and the inlet temperature is 12°C. Mass flow rate of water taken 0.05 m/sec

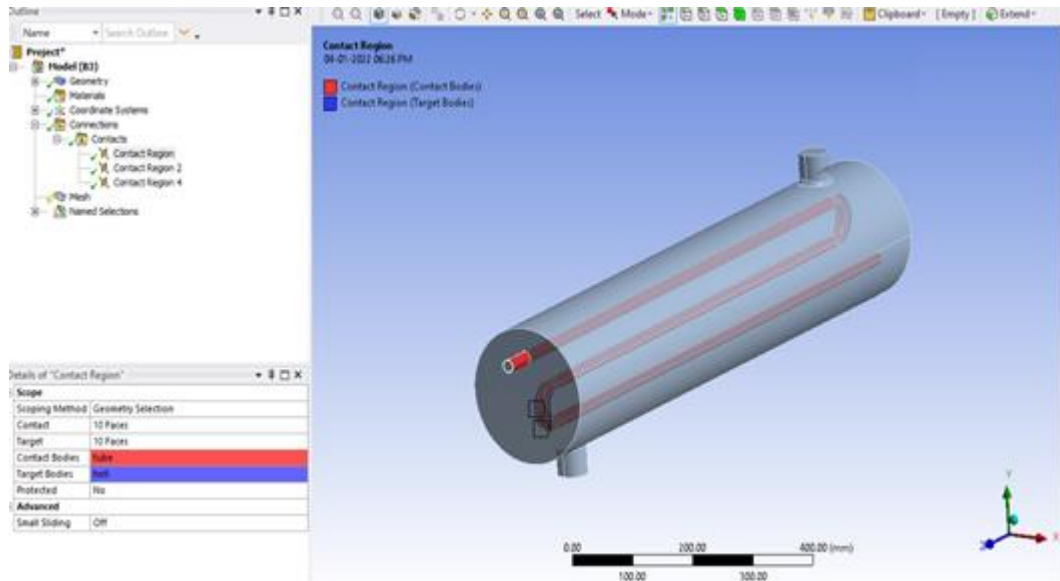


Figure 5.2: 'S' Pattern tube used inside heat exchanger model

➤ **Case 3**

A heat exchanger with a helical tube is employed in Example 3. The diagram depicts a Shell and Tube Heat Exchanger with Helical Type Tubes. The shell of the exchanger receives hot water at a temperature of 85°C , and helical type tubes flow through it. The tube is filled with cold water and the cold water intake is set at 18°C . Mass flow rate of water taken 0.05 m/sec

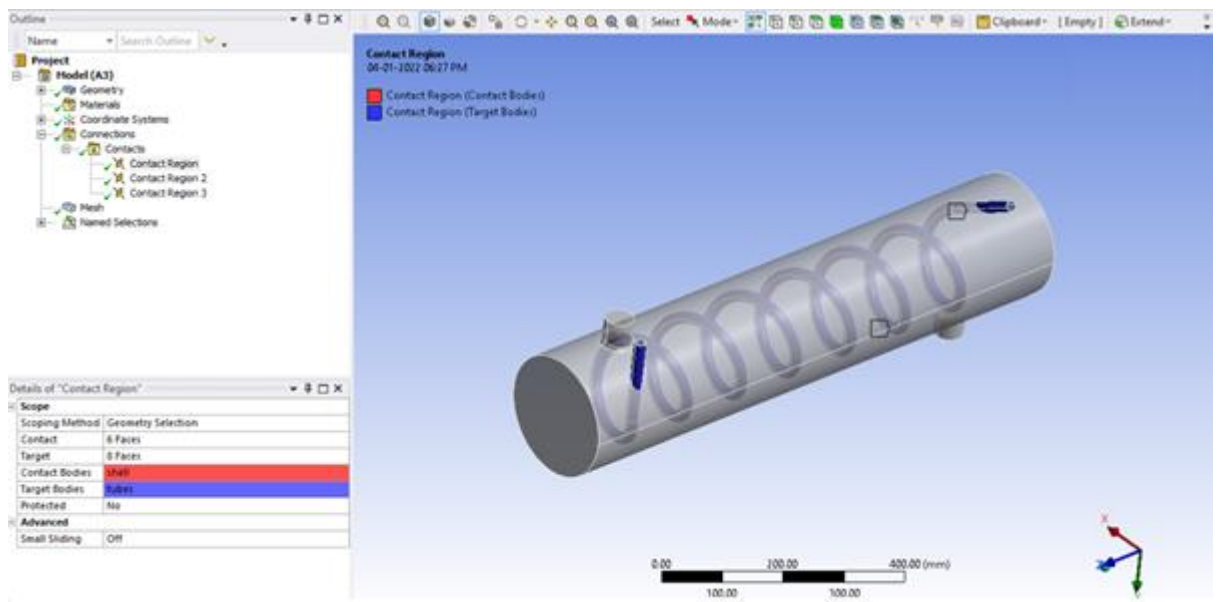


Figure 5.3: Helical type Tube used inside Heat Exchanger

➤ Case 4

The heat exchanger in Case 4 is constructed using a Zigzag pattern tube. The Shell and Tube Heat Exchanger with Zigzag Pattern Tubes is shown in the diagram. The entrance temperature of hot water is 85°C for the shell of the exchanger, and Zigzag pattern type tubes flow through the shell. The tube is filled with cold water, and the temperature of the cold water intake is 18°C . Mass flow rate of water taken $0.05\text{ m}^3/\text{sec}$

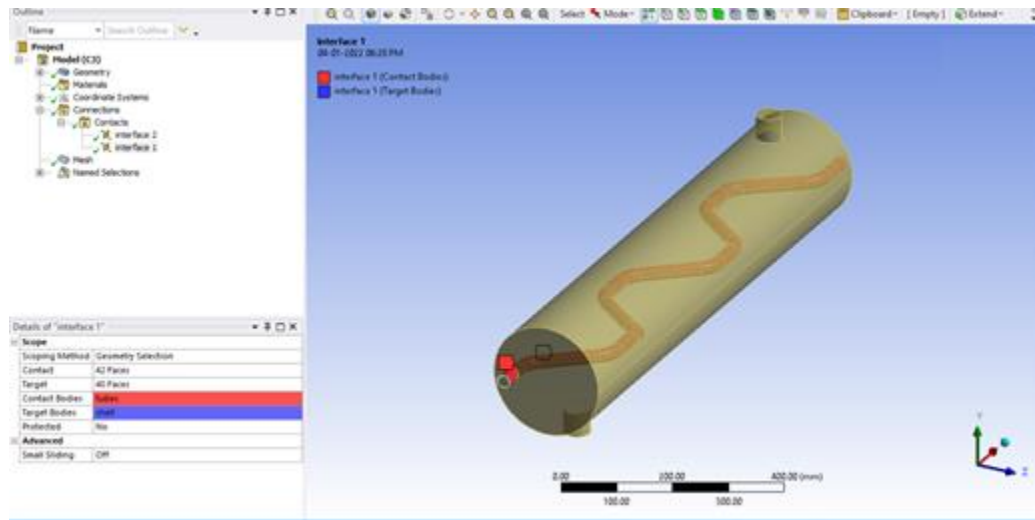
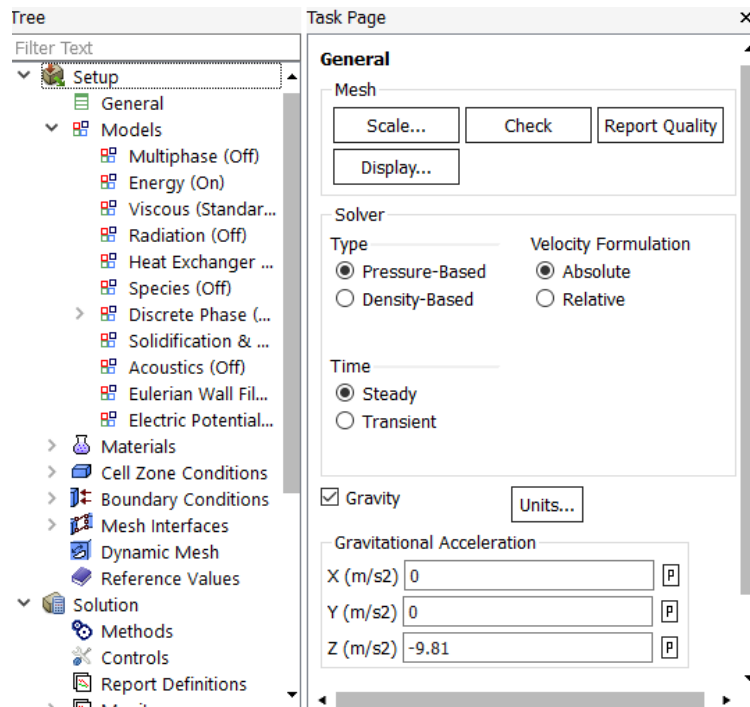


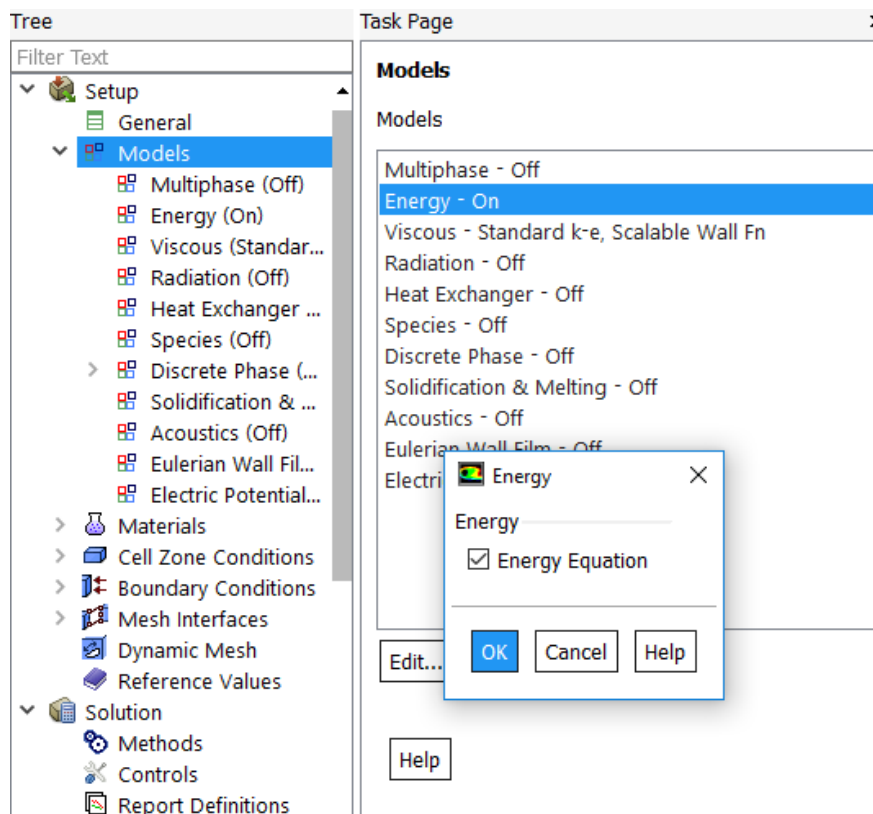
Figure 5.4: Zigzag type Tube used inside heat exchanger model

5.4. Set up of ANSYS software

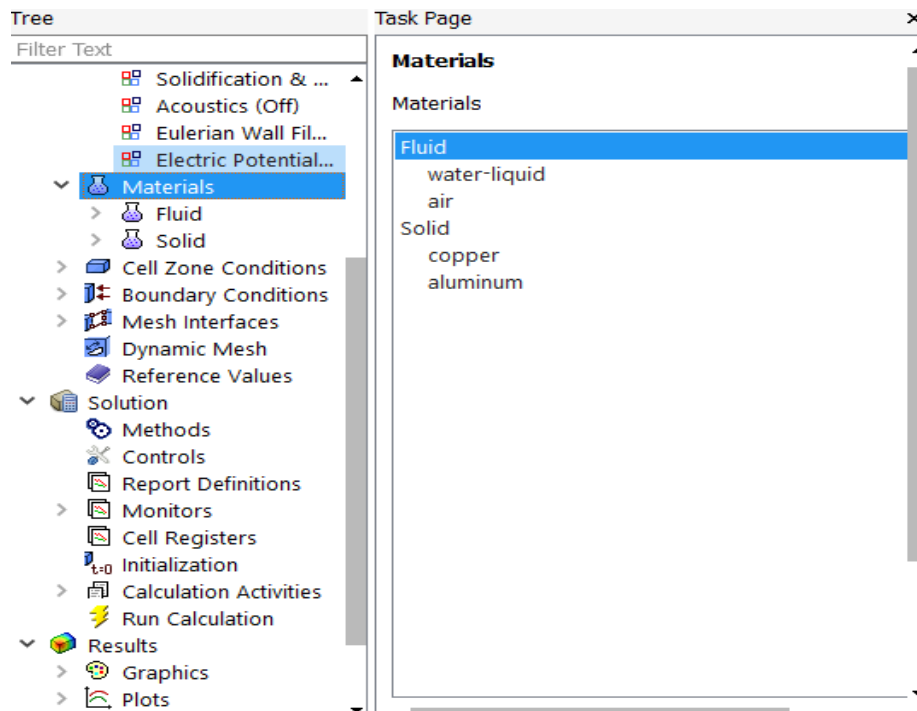
a. Initial step in CFD Fluent



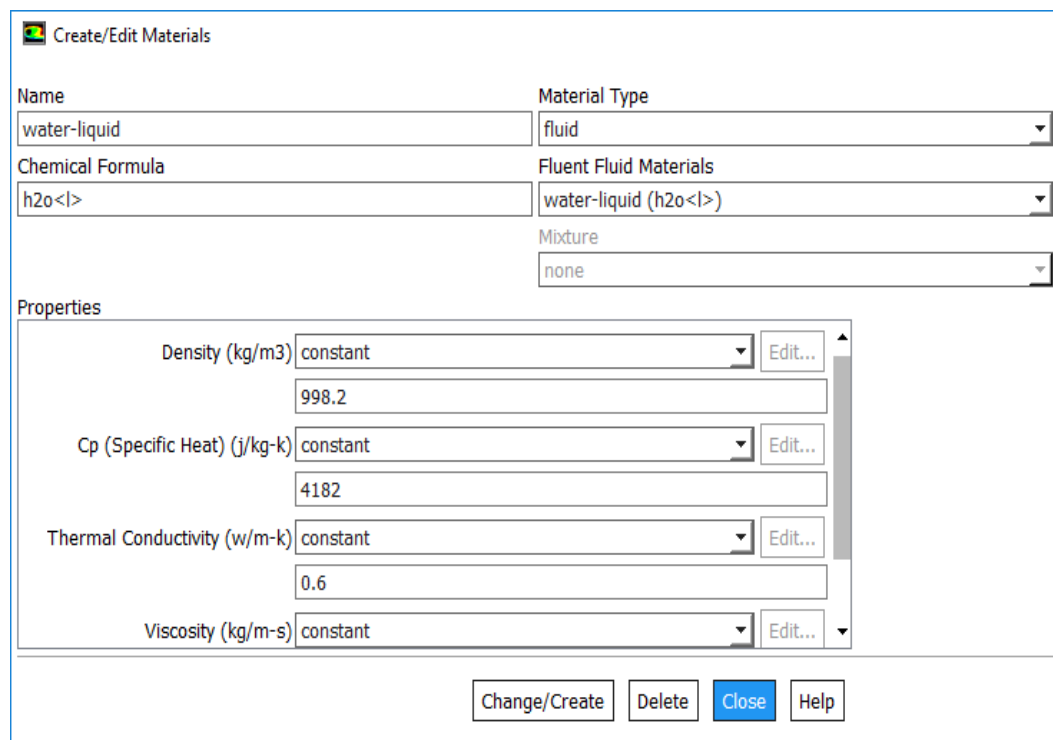
b. Model specification is the next stage following generalization.



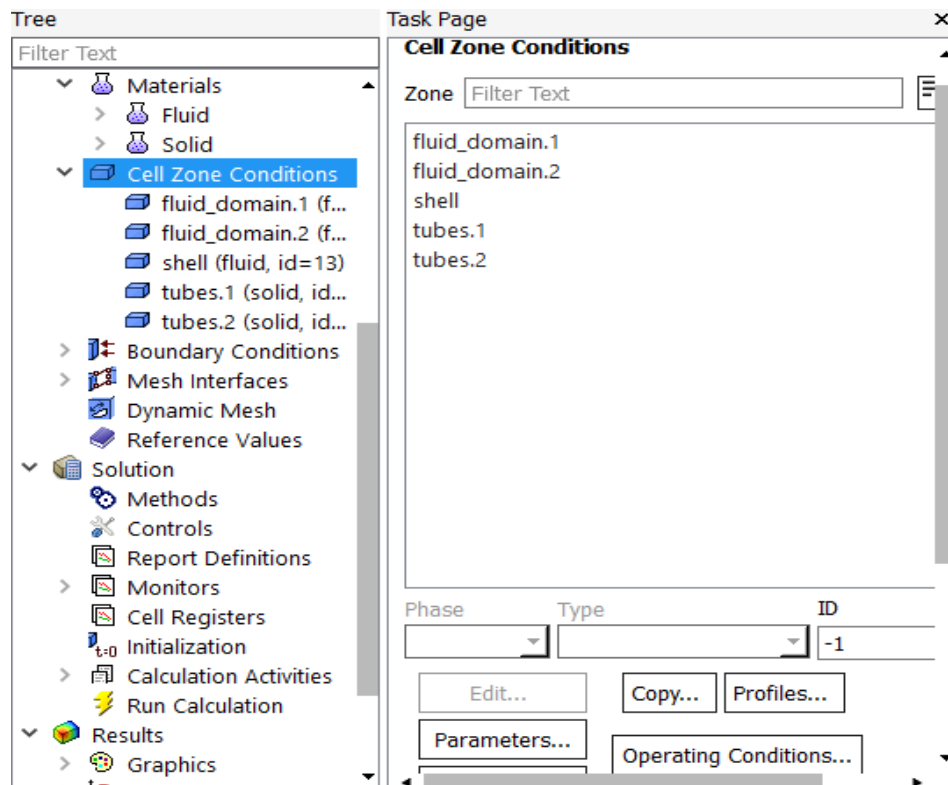
c. The next step is to pick the right materials: Inserting material that we'll be using in our research.



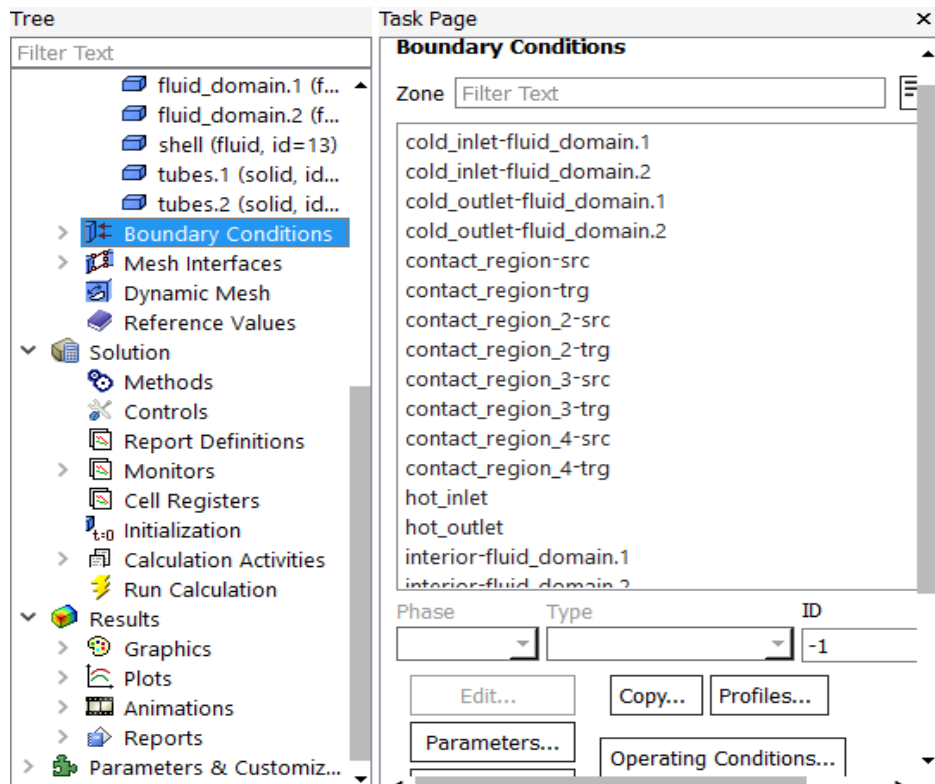
d. Using a fluent database to edit Air properties.



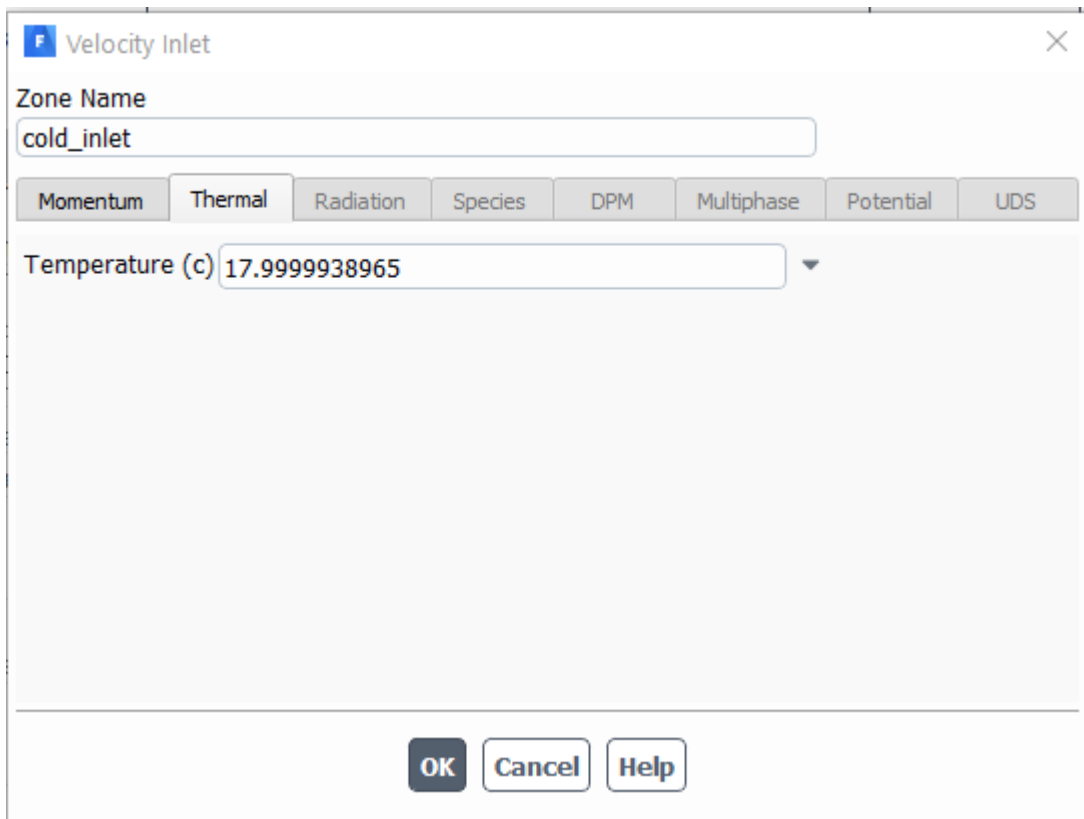
e. Parameterization of cell zones



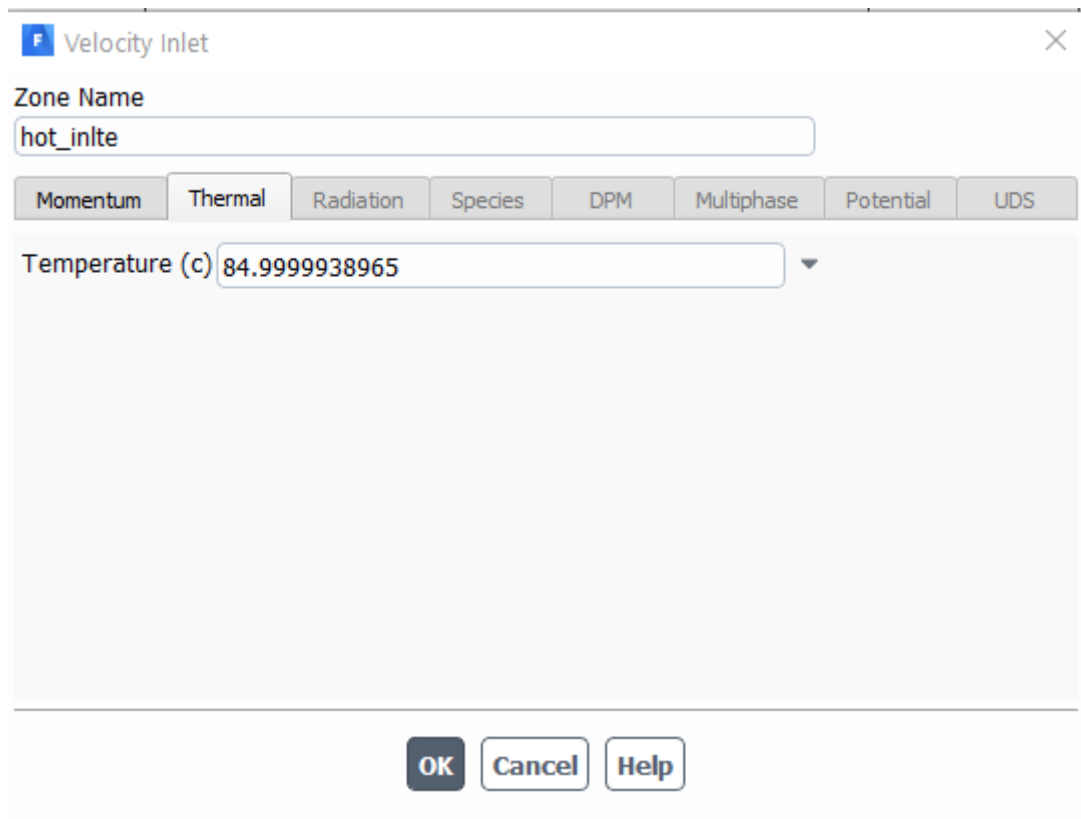
f. Setup of Boundary conditions



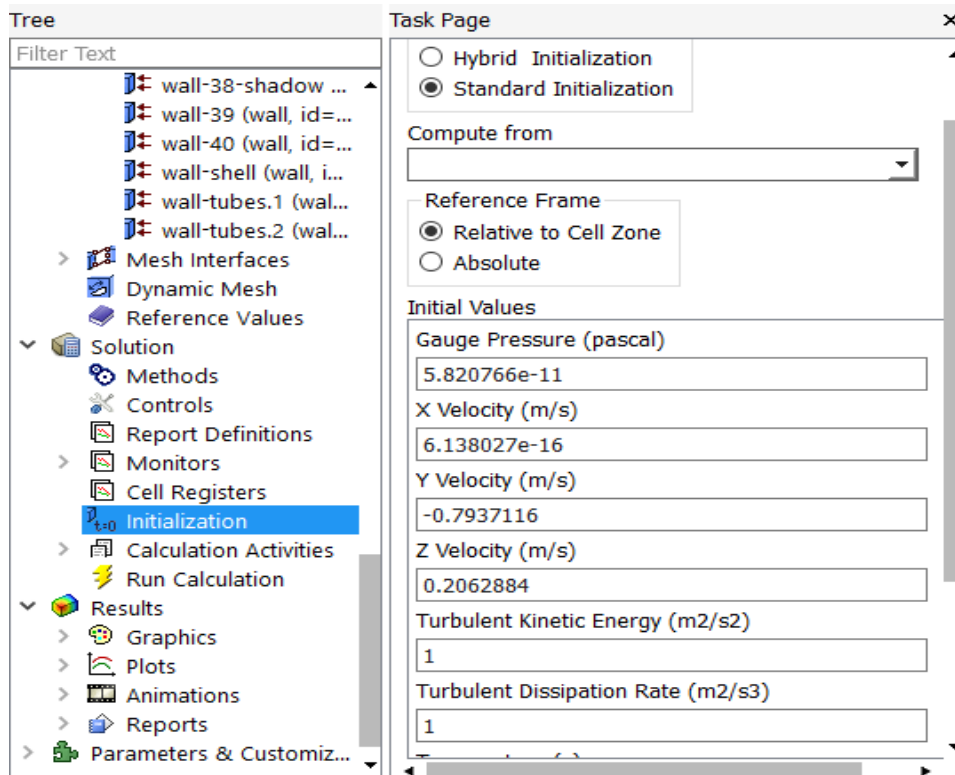
g. Insert Cold Fluid temperature



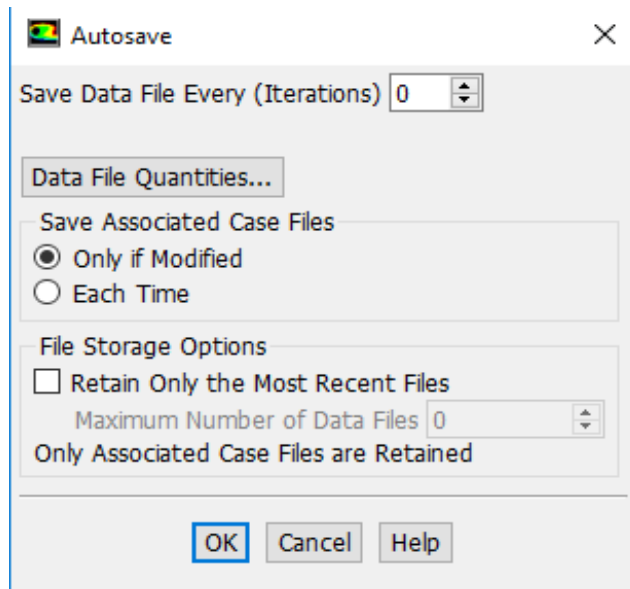
h. Insert Hot Fluid temperature



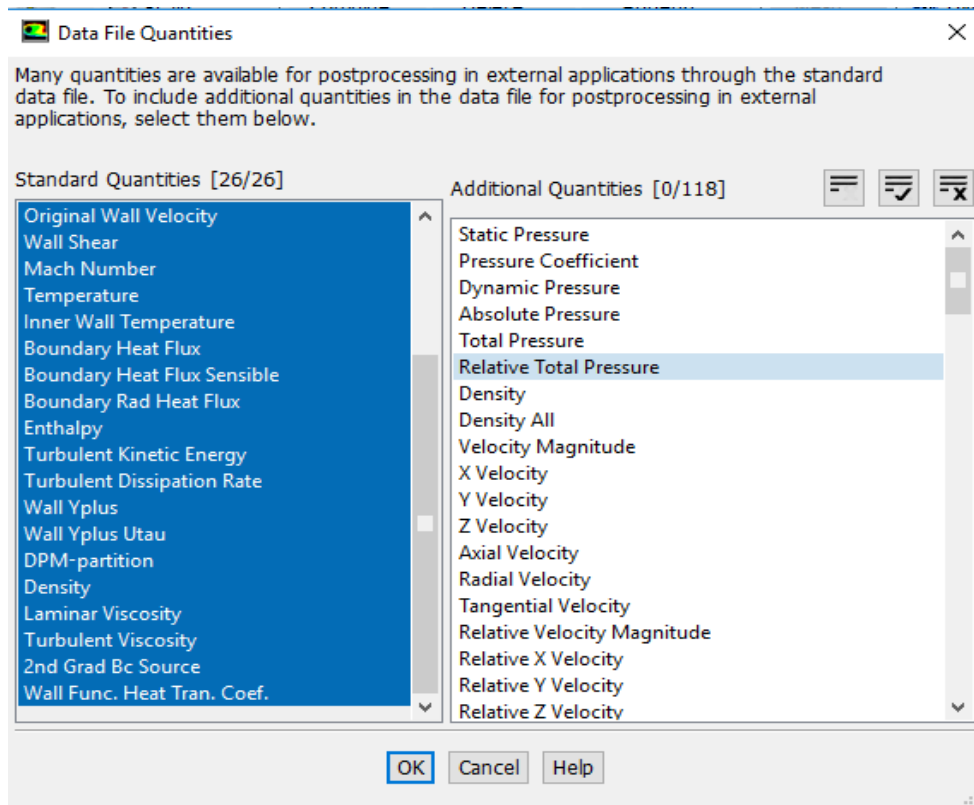
i. Run Initialization solution setup



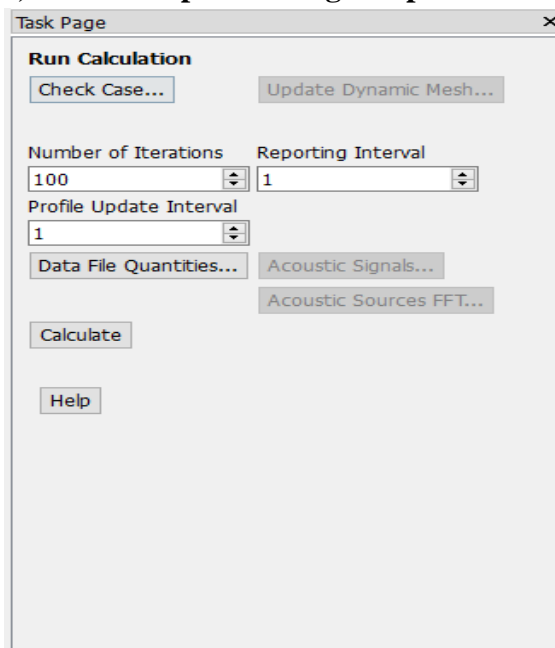
j. Sava data file for analysis



j. Choosing the data we'll be looking at throughout our analysis



k. For our analysis, we've set up a running computation.



CHAPTER 6

RESULTS AND DISCUSSION

CHAPTER 6

RESULTS AND DISCUSSIONS

6.1 Introduction

In this chapter, four case studies of Shell and Tube Heat Exchangers are discussed, the first of which is a parallel flow tube heat exchanger. The heat exchanger in Case 2 is constructed using a 'S' Pattern type flow tube. Heat exchangers with helical tubes are used in case 3 while heat exchangers with zigzag pattern tubes are used in instance 4. The entrance temperature of hot water is 85°C for the shell of the exchanger, and helical type tubes flow through the shell. The tube is filled with cold water, and the temperature of the cold water intake is 18°C.

6.2. Results of Heat Exchanger

By running a CFD simulation of all three types of heat exchangers using various tubes, we were able to determine the intake and exit temperatures.

6.2.1. The temperature of cold water at the tubes' output was investigated.

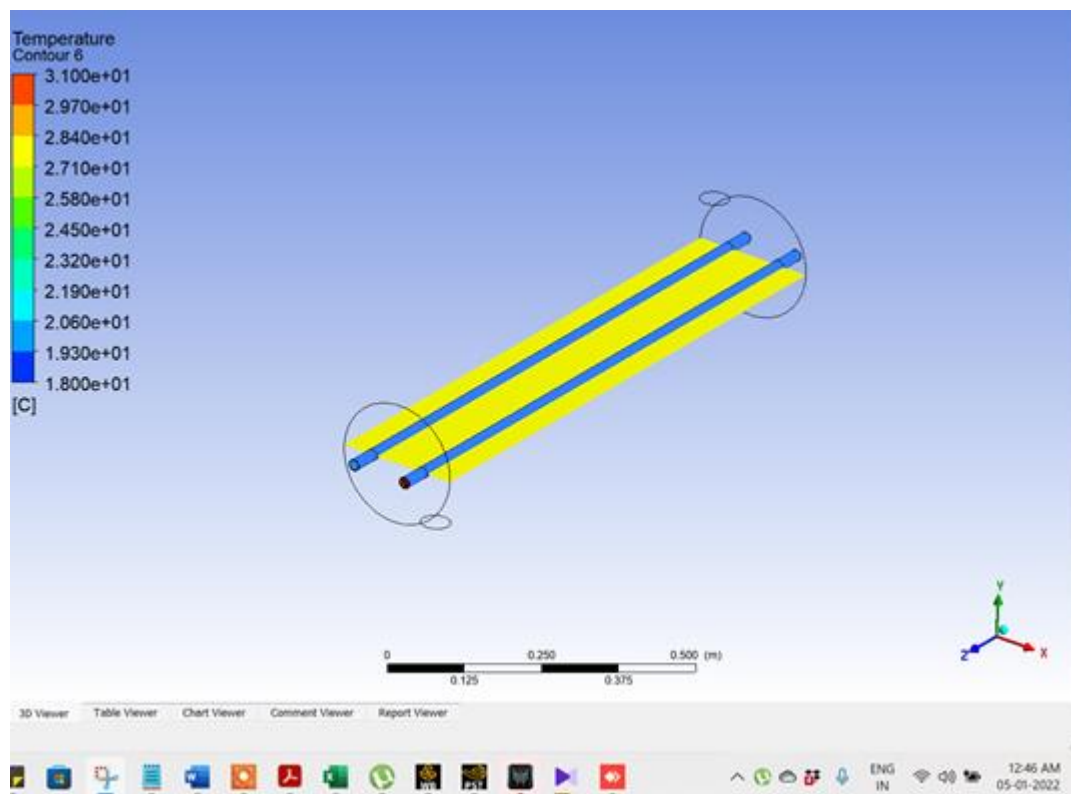


Figure 6.1: Temperature fluctuations in parallel pipe flows

“The temperature fluctuations of a shell and tube heat exchanger model with a straight tube design are shown in Figure 6.1, where the maximum temperature of cold fluid is discovered”.

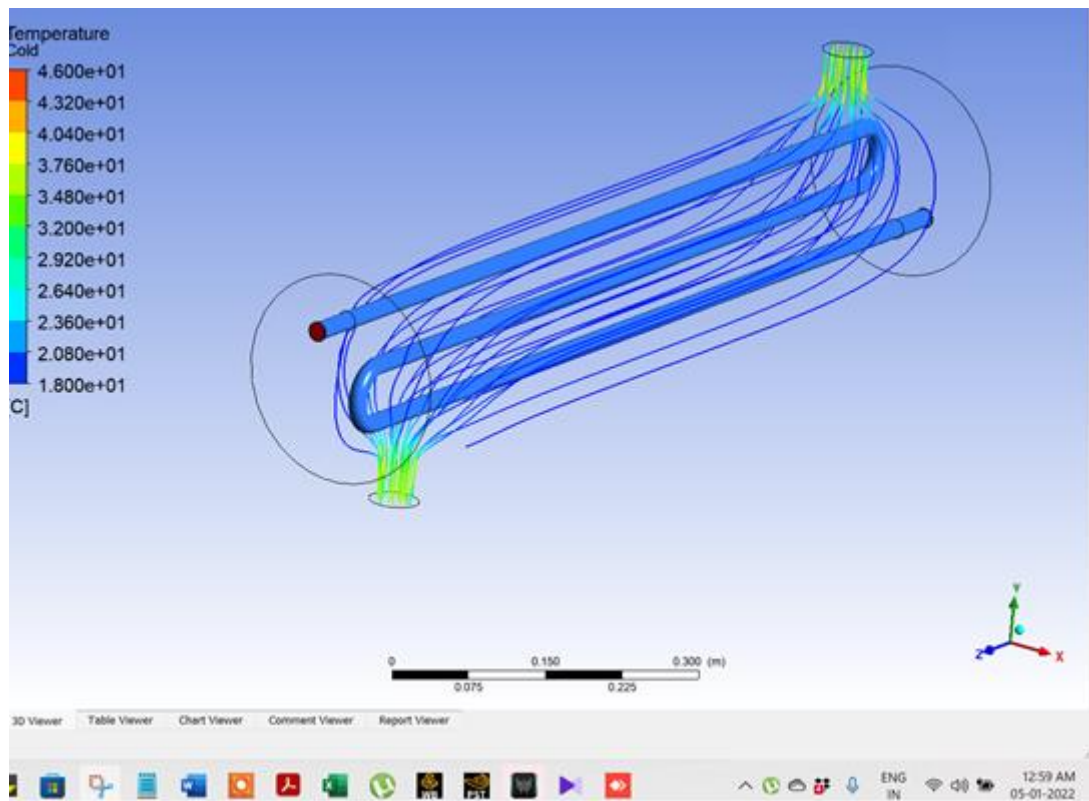


Figure 6.2: Temperature changes in the 'S' Pattern pipe flow

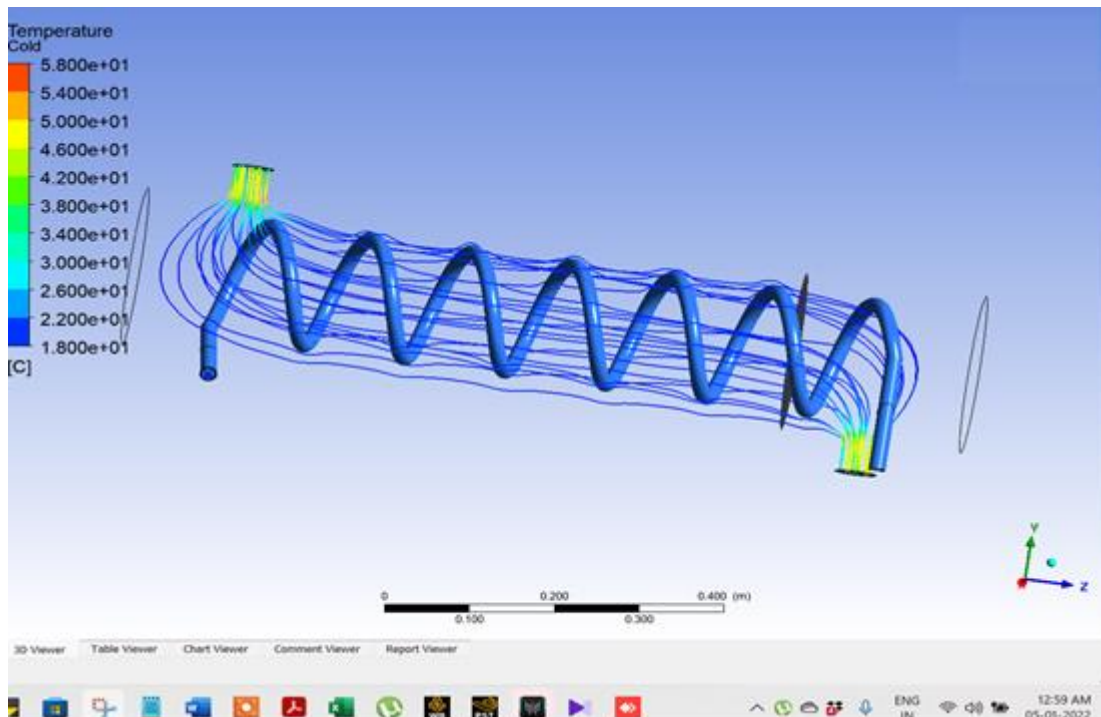


Figure 6.3: temperature fluctuations in helical pipe flow

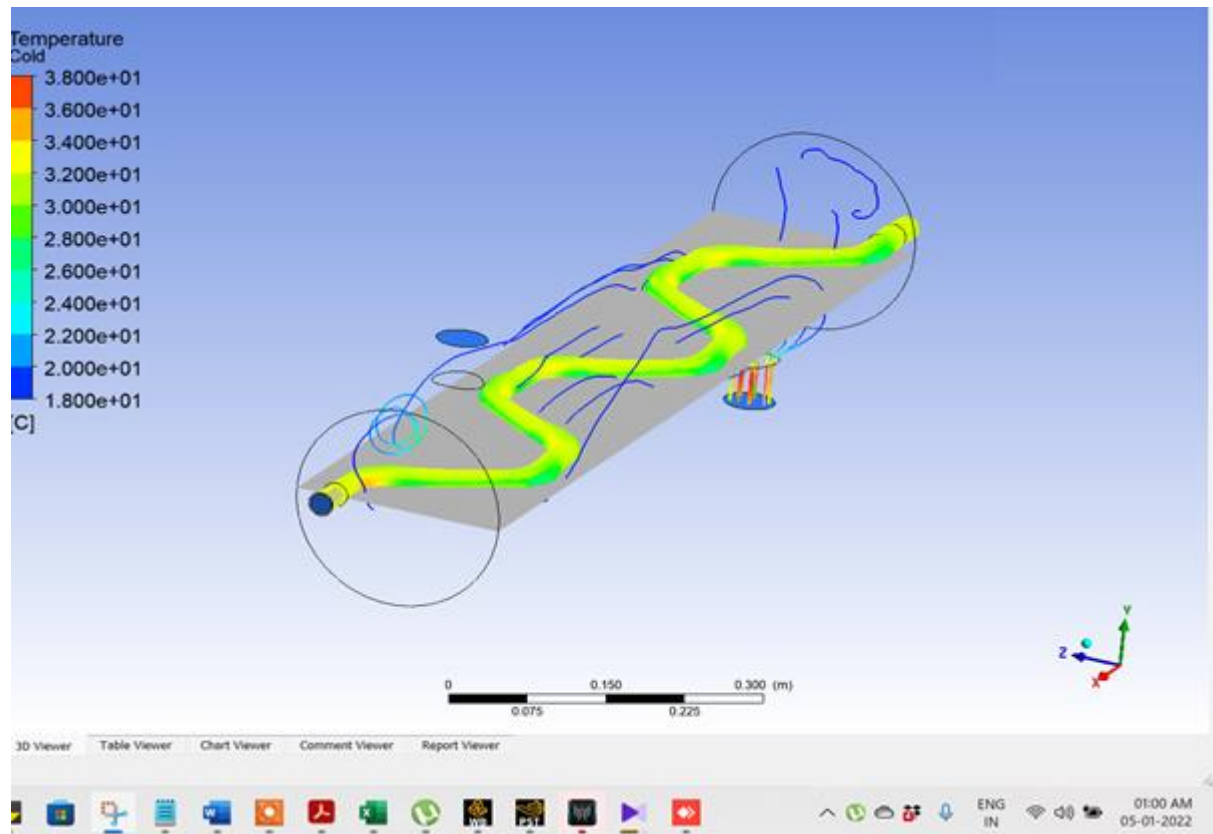


Figure 6.4: Pipe flow temperature changes in a zigzag pattern

The temperature differential changes with time, as seen in the figure. Ansys was used to create three different kinds of heat exchangers in the research mentioned above. Parallel, 'S,' zigzag, and helical heat exchanger tubes are only a few of the forms and patterns available. The intake temperature of 18°C and the input temperature of 85°C were determined after the design phase. CFD analysis was applied to enhance heat transport in the heat exchanger design. Mass flow rate of water taken 0.05 m/sec

Table 6.1: Temperatures of cold-water flow at the intake and outflow

Pattern of Tubes	Cold water inlet (°C)	Cold water outlet (°C)
Parallel tubes	18	31
'S' pattern tubes	18	46
Zig zag pattern tube	18	38
Helical tubes	18	58

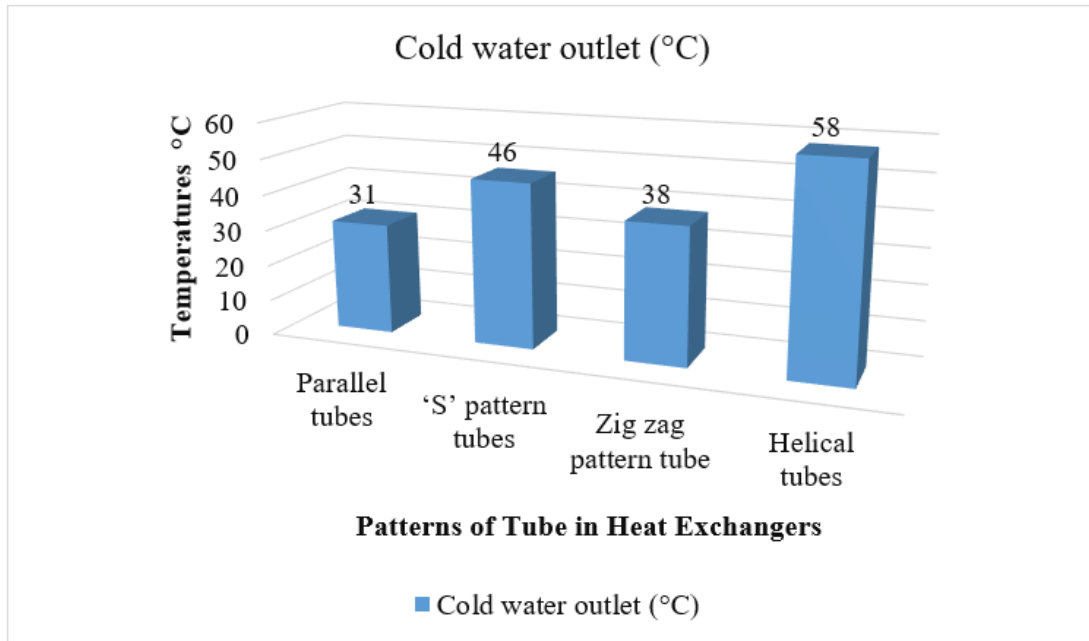


Figure 6.5: Temperatures of cold water in tube heat exchangers

According to a graph and academic studies, the lowest heat exchange rate is seen in parallel tube heat exchangers. “The temperature of cold water enters a parallel tube heat exchanger at 18°C and departs at 31°C. In comparison to a Parallel tube heat exchanger, This heat exchanger has a lower 18°C input temperature and a higher 46°C exit temperature”. The entry temperature of parallel tube heat exchangers is 18°C, whereas the exit temperature is only 58°C.

6.2.2. Study of hot water analysis at outlet

Hot water flows inside the shell of all of the models with varied tube forms that were created. Following a simulation, the final temperature values are recorded and compared to other models' temperatures, and a graph is displayed to help people understand how the suggested model works.

The hot water temperature at the shell's exit is seen in Figures 6.6 to 6.9.

As seen in Figure 6.6, the suggested model of a shell and tube heat exchanger in which cold water is cycled through the tube and hot water is pumped through the shell is displayed. For hot water, the tubes have an input temperature of 85°C, and for cold water, the temperature is 18°C. The temperature decrease of hot water from the highest to the lowest temperature in a straight tube pattern heat exchanger model is 84°C, as indicated in Figure 6.6.

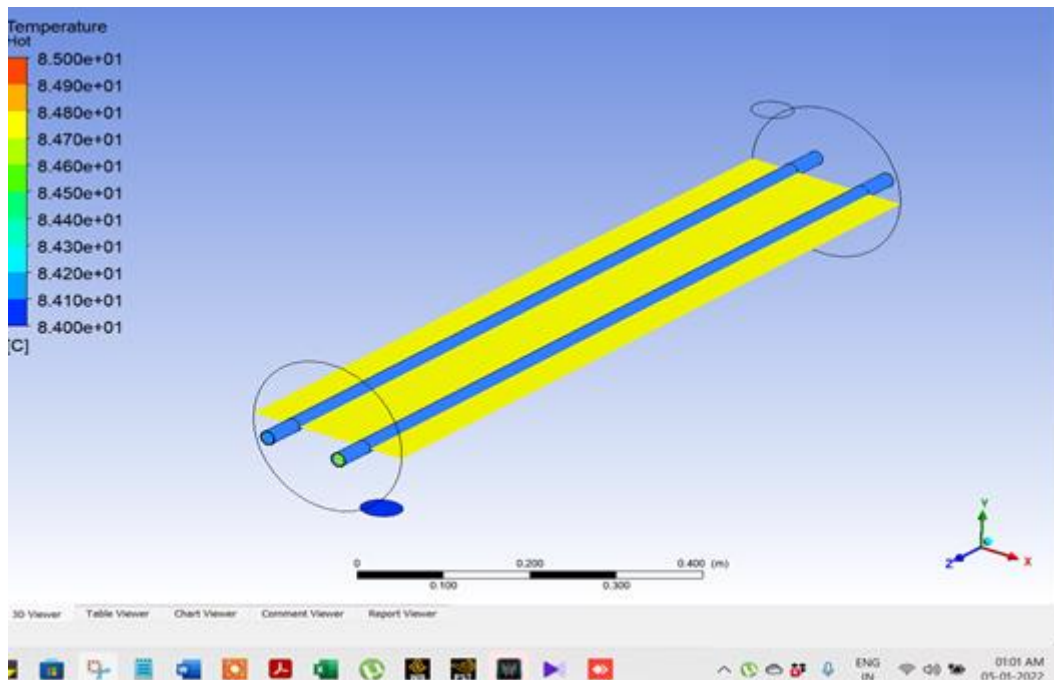
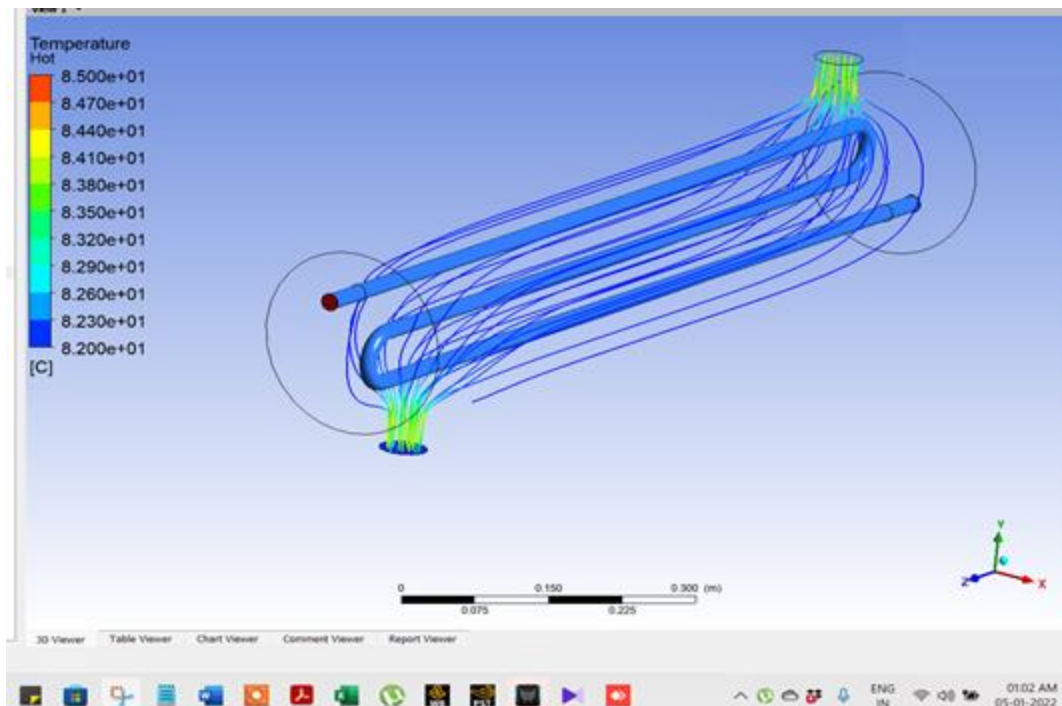
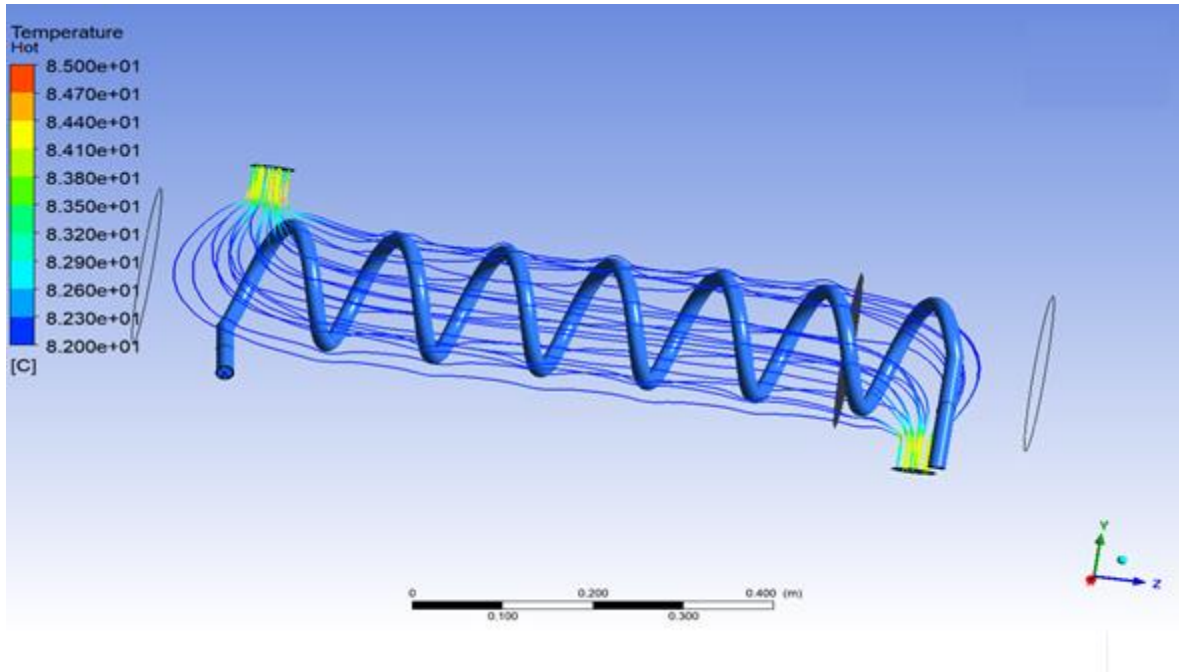


Figure 6.6: Temperature of hot water from the shell at the parallel tube's output
 A model of a heat exchanger with a 'S' design is shown in Figure 6.7. Two u tubes are joined together in this model to extend the tube's length, as seen in the diagram below. This hot water model's maximum temperature drop is from 85°C to 82°C.



“Figure 6.7: Temperature of hot water from the shell at the 'S' Pattern tube's output”



“Figure 6.8: Temperature of hot water from the shell at the tube's outflow in a helical pattern”

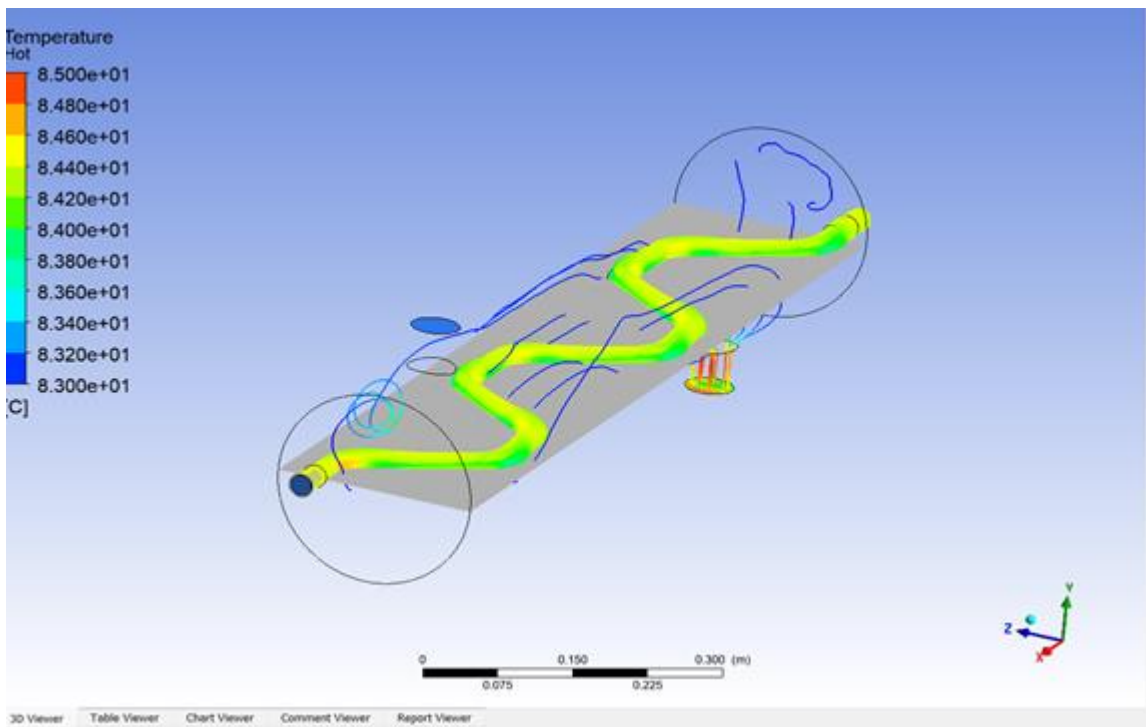


Figure 6.9: Temperature of hot water from the shell at the tube's outflow in a zigzag pattern

Figure 6.9 In a zigzag pattern tube type heat exchanger, the maximum hot temperature was reduced from 85°C to 83°C. Table 6.2 shows the fluctuations in hot water temperature at the shell's output. The shell's hot input water temperature is 85°C, which is consistent across

all models. After that, the output temperature was measured and compared to all of the model temperatures. Mass flow rate of water taken 0.05 m/sec

Table 6.2: Temperatures of hot water at the inflow and outflow

Pattern of Tubes	Hot inlet (°C)	Hot outlet (°C)
Parallel tubes	85	84
'S' pattern tubes	85	82
Zig zag pattern tube	85	83
Helical tubes	85	82

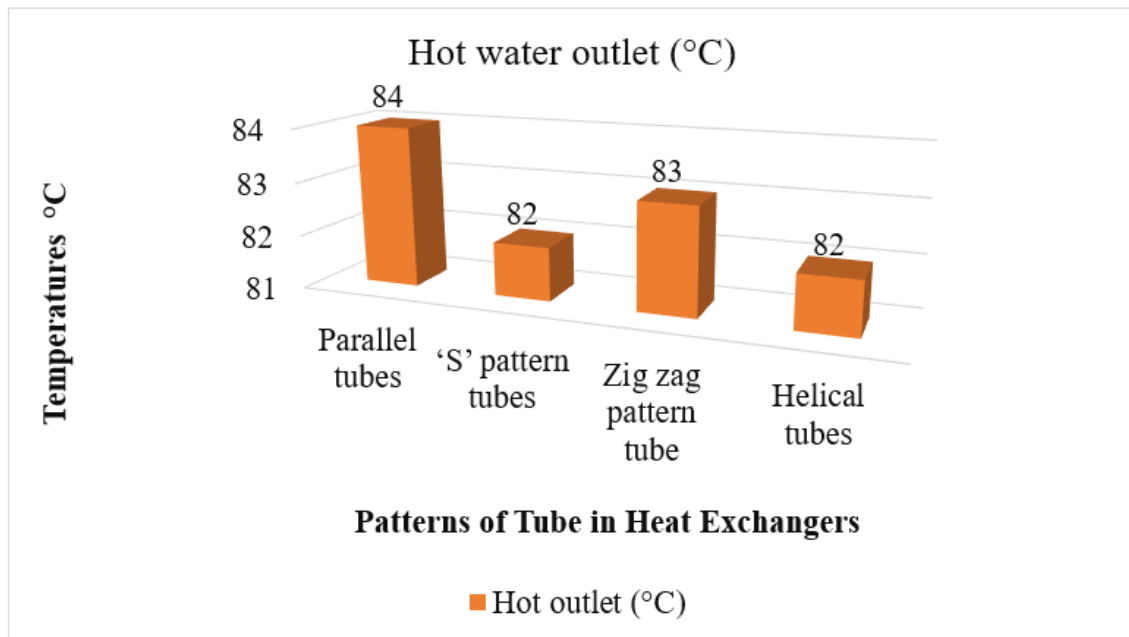


Figure 6.10: Hot water temperatures in heat exchangers dependent on tube layout

The best approach to heat things up is regarded to be a series of parallel tubes. Heated water enters a parallel tube heat exchanger at 85 degrees Celsius and exits at 84 degrees Celsius. People who use "U" Pattern tube heat exchangers can remove more heat more quickly than people who use Parallel tubes with an input temperature of 85°C and an output temperature of 82°C. Helical tube heat exchangers provide a 3°C temperature decrease with 85°C hot water input and 82°C hot water output.

6.3. Velocity of Tubes at outlet

The outlet velocity of each designed model of a Shell and Tube heat exchanger was evaluated. Because varied forms of tubes create velocity changes, it was important to study the outlet velocity of each designed model. I estimated the velocity using the Ansys simulation and compared it to the model behaviour as a function of form changes. The tube exit velocity is shown in Table 6.3.

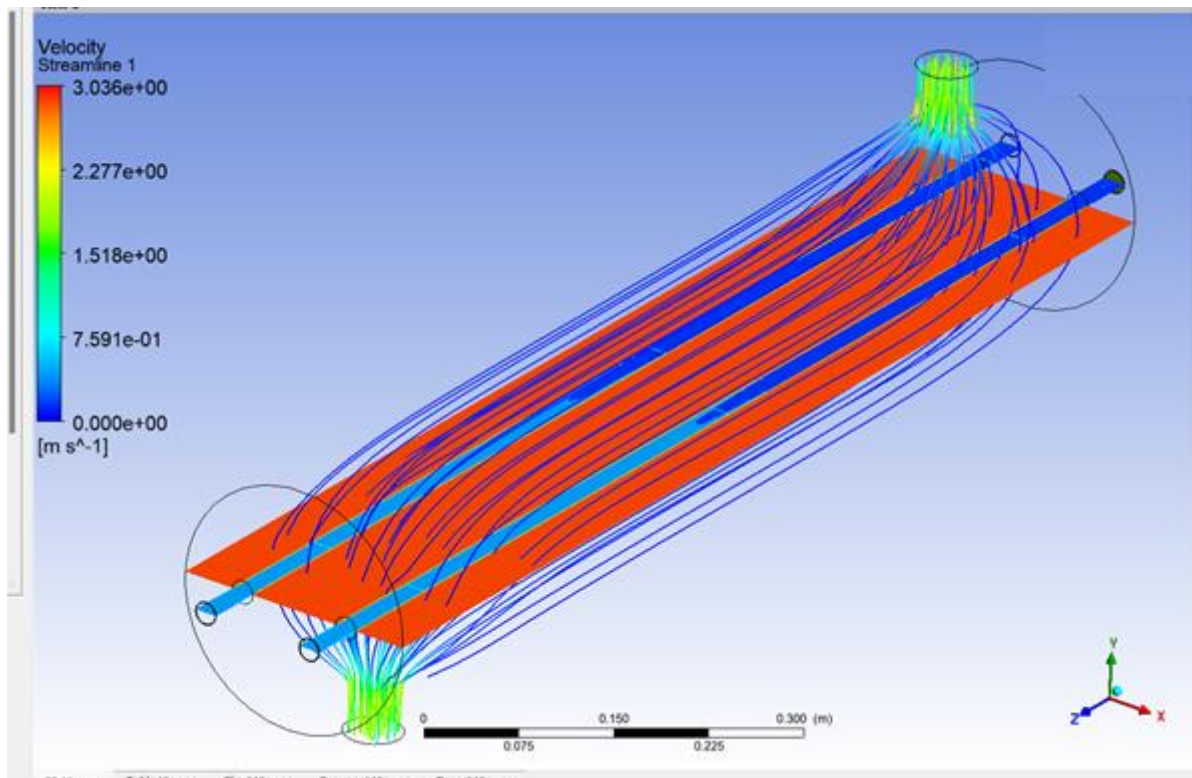


Figure 6.11: Flow rate of a parallel tube pipe Streamline

Figure 6.11 illustrates the velocity streamline of a straight parallel tube, which was developed to examine heat transfer in a constructed model. The tube's entrance velocity was 1.85 m/s, and the output velocity was 3.036 m/s. In the straight tube pattern, it indicates an increase in velocity.

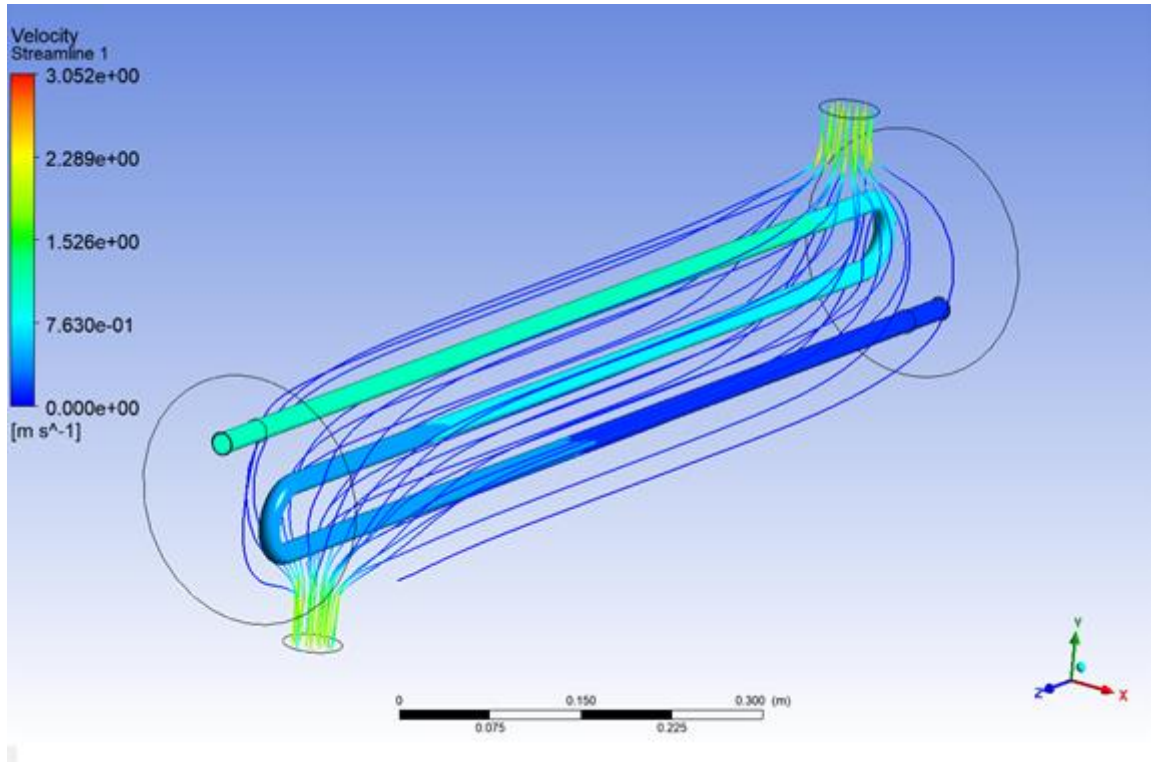


Figure 6.12: Tube pipe flow velocity Streamline 'S' Pattern

Figure 6.12 shows the velocity and temperature at the tube's outlet as a result of a 'S' pattern generated in the tube's exit. Findings from a simulated investigation showed that the output velocity was 3.052 metres per second (m/s) and the intake velocity was 1.85 metres per second.

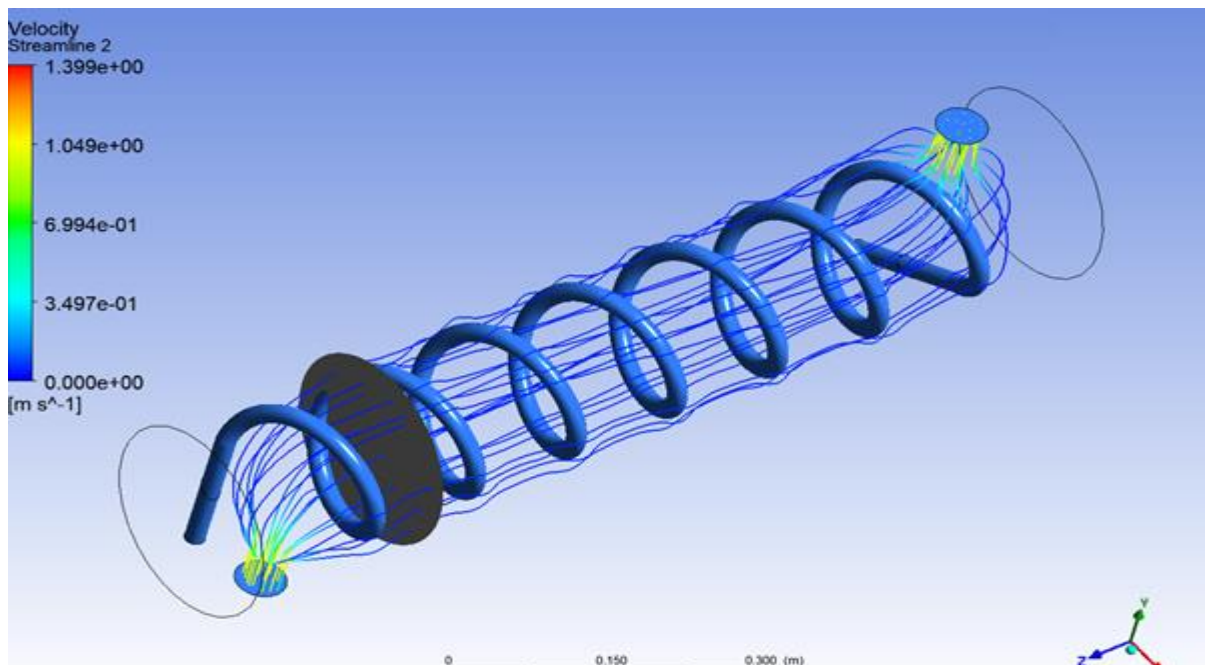


Figure 6.13: streamline velocity flow in a helical tube and a shell

Figure 6.13 depicts the tube's outlet velocity streamline; in this model, I made the tube helical to analyse the outlet velocity and temperature; simulation analysis showed that the outlet velocity was 1.399 m/s and the input velocity was 1.85 m/s.

Heat exchangers with zigzag pattern tubes are shown in Figure 6.14. When cold and hot liquids are exchanged in a tube, I used a zigzag pattern to observe how temperature and velocity change at the output.

According to this model, the water's mean velocity is 1.85 m/s, which means the zigzag tube has a maximum velocity of 2.57 m/s.

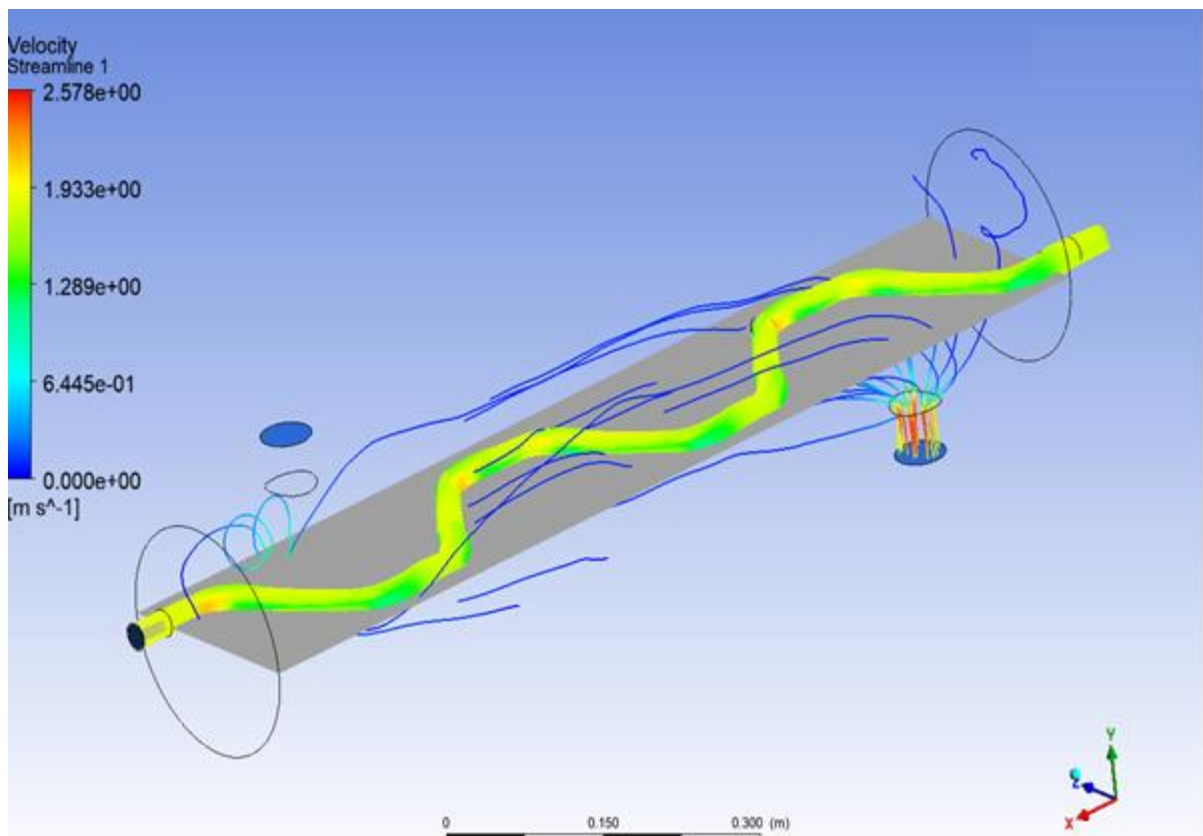


Figure 6.14: The zigzag tube and shell combine to create a high-velocity flow.

In this table, I've compared the flow rate from the input to the exit of the heat exchanger as a function of various tube shapes, in order to examine the effect of tube shape modifications on the output water temperature. Mass flow rate of water taken 0.05 m/sec

Table 6.3: Comparison of Tube Velocity at Outlet

Pattern of Tubes	Velocity at tubes Inlet [m/s]	Velocity at tubes Outlet [m/s]
Parallel tubes	1.85	3.036
'S' pattern tubes	1.85	3.052
Zig zag pattern tube	1.85	2.57
Helical tubes	1.85	1.399

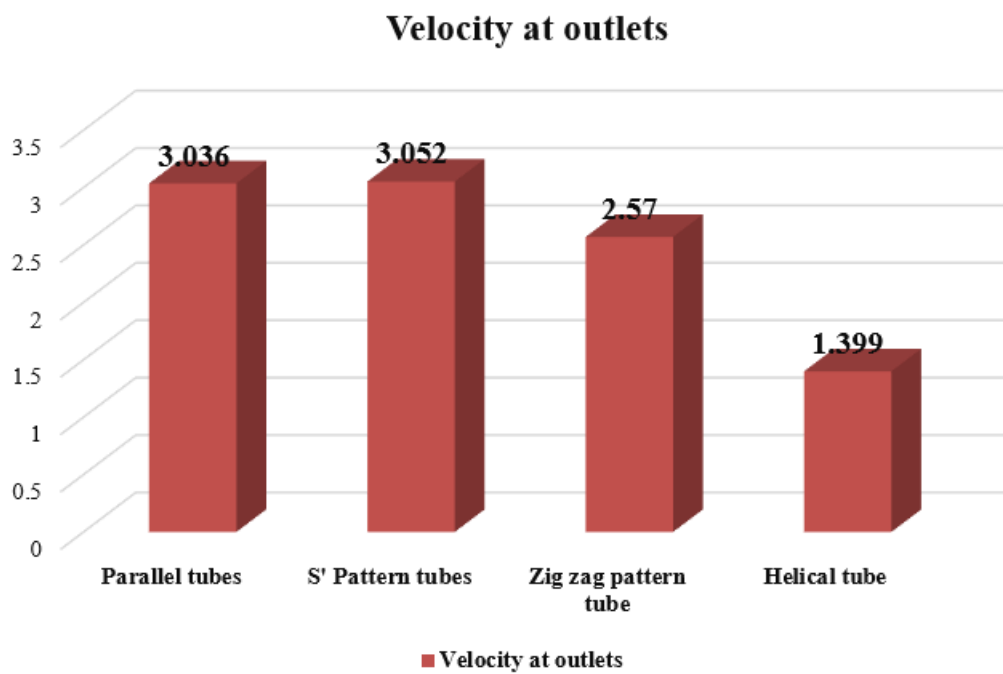


Figure 6.15: Variations of Velocity at outlets of tubes

6.4. Data validation of Research work

Thermal Performance of Solar Water Heater by Using Ansys Fluent

Manisha Jauhri, Pooran Singh Dhakar, "Thermal Performance of Solar Water Heater by Using Ansys Fluent", IJRAR, Volume 6, Issue 2, June 2019.

https://www.researchgate.net/publication/345002249_Thermal_Performance_of_Solar_Water_Heater_by_Using_Ansys_Fluent

Results of outlet waters from solar water heater (Manisha et. Al. 2019)

Solar Water Heater Design	Inlet Temperature (°C)	Maximum outlet Temperature Rise in (°C)	Maximum Pressure Rise in (Pa)	Maximum Velocity (m/s)
Vertical Straight Fins	27	39	4.06	0.0836
Vertical Curved Fins	27	67	4.22	0.0898

For my work research data validation, I have selected this paper in which solar water model designed with variations in plates of fins which are attached in side the box of solar water heater model. I have designed only one model for data validation, model with vertical straight fins.

All dimensions are taken from the base paper of Manisha et. Al. 2019;

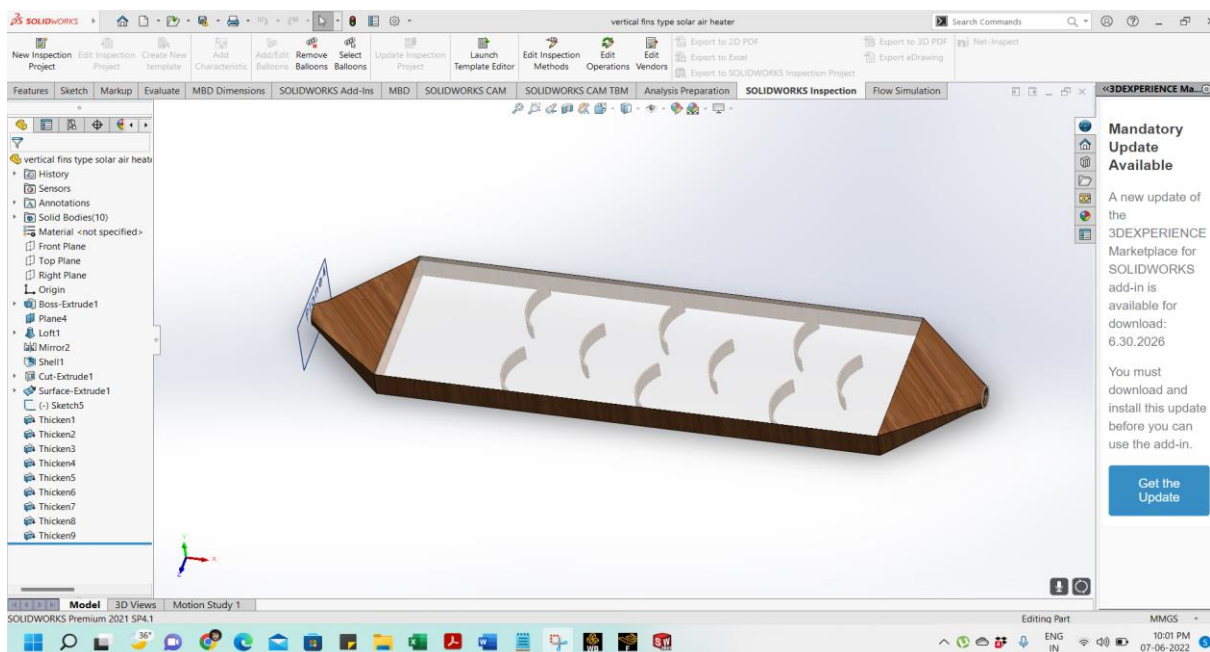


Figure 6.16: Solar water model designed in Solidworks

Solar water model imported in Ansys for analysis.

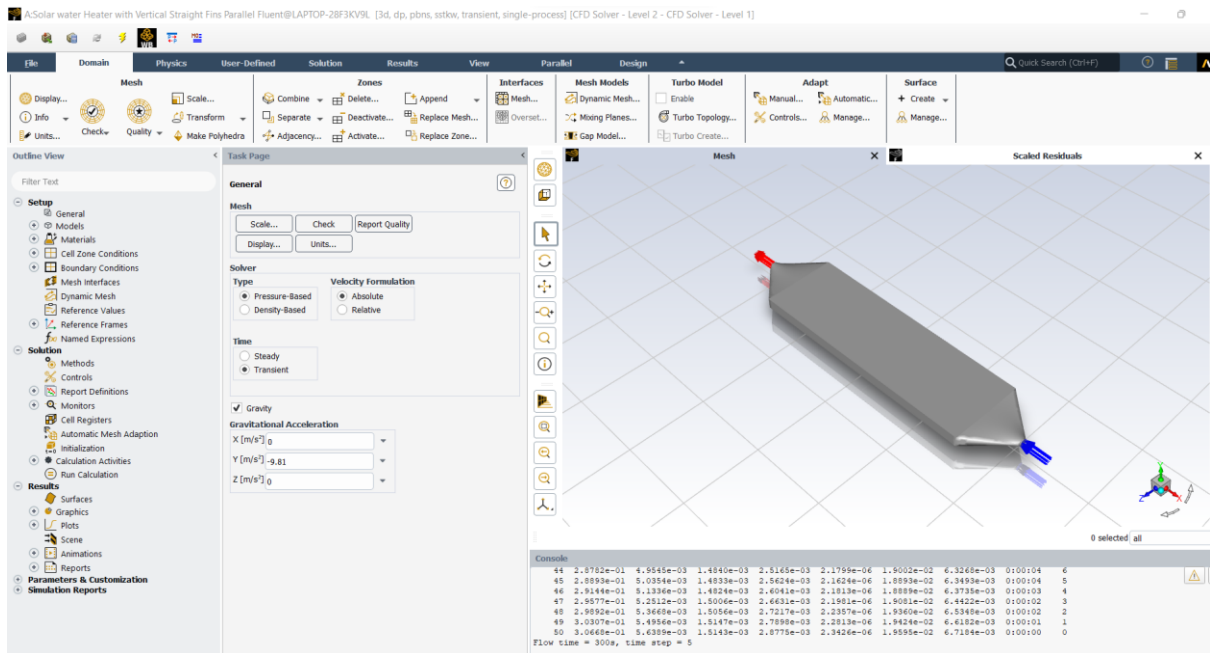


Figure 6.17: Solar water model in Ansys

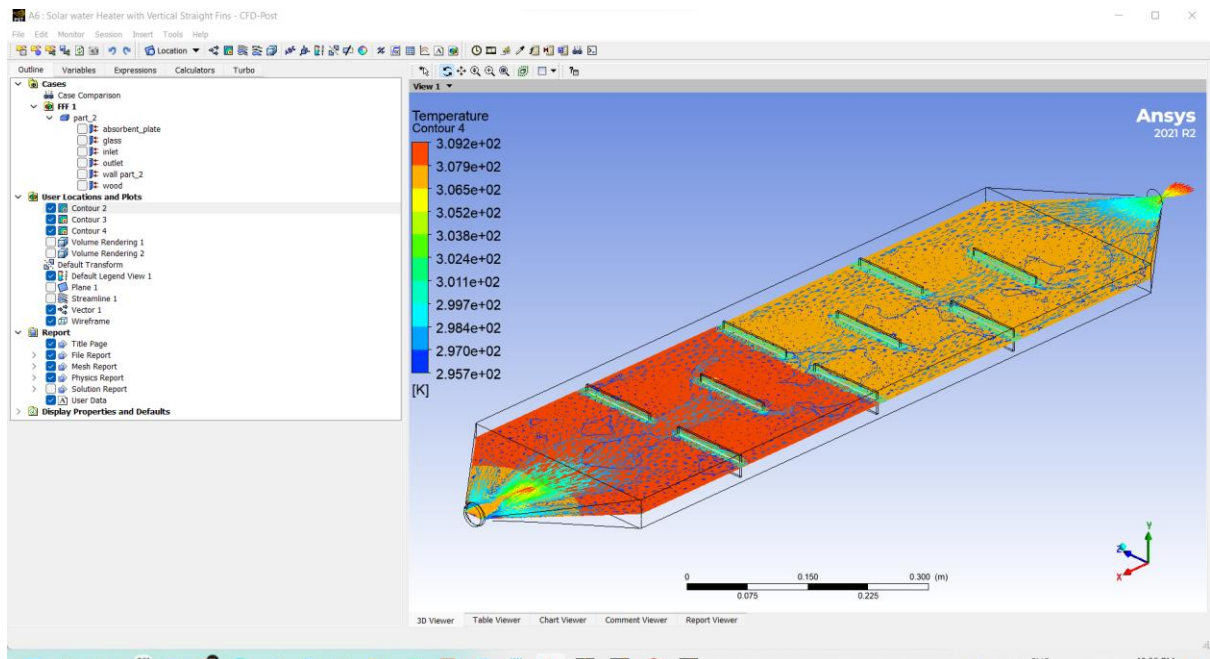


Figure 6.18: Temperature at outlet in Solar water model

6.5. Data Table:

Data found in our analysis Outlet temperature of water

Our Simulation Data	Literature Data
	Manisha et. Al. 2019;
Solar water heater model with vertical fins	Solar water heater model with vertical fins
Outlet Temperature (Max)	Outlet Temperature (Max)
36.05 °C	39 °C

$$\text{Percentage (\%) of variations} = \frac{(\text{Expriment value} - \text{Literature value})}{\text{Expriment value}} \times 100$$

$$= \frac{(36.05 - 39)}{39} \times 100 = - 8.19 \% \text{ (For Maximum Temperature)}$$

CHAPTER 7

CONCLUSION

7.1. Conclusion

To optimize how Heat Exchangers may be used, a comprehensive data analysis was performed on the three various tube variations present in Shell and Tube Heat Exchangers, and the findings were compared to determine the optimum research to exchange heat.

Heat exchangers made of shell and tube have been the subject of several studies and experiments. Heat exchangers are designed using 3-D finite element modelling of classic Shell and Tube designs. As a result of employing FLUENT in a CFD study, the temperature of the hot water dropped according to the volume. ANSYS V14.0 is used for modelling, meshing, preprocessing, and analysis.

- Analyzing and adjusting the shell design using Ansys was used for this project to get the greatest potential temperature fluctuation. Steel was utilized in the shell and aluminum tubes were used in the tubes. Heat exchange between the shell and its surroundings is currently regarded to be ineffective. In terms of heat transfer efficiency, the heat exchanger's S-pattern tube design is superior than other designs.
- From 18°C to 31°C, the temperature of the outflow is raised by using parallel flow tubes. In comparison, the water exiting the 'U' pattern tubes is 46°C, whereas the water exiting the Helical pattern tubes is 58°C. After comparing the results, the helical tube type design model and the 'U' pattern heat exchanger were determined to have the best heat transmission. The heat transfer efficiency of a heat exchanger with helical tubes is the best.
- The velocity of helical tubes in shell and tube heat exchanger types is 3.052 m/s, which is greater than the other versions, according to a study. The lowest velocity was found on the helical tube heat exchanger model.

7.2. Future Scope

- To complete the experiment, several tube profiles on the heat exchanger shell and various nano fluids may be employed.
- It is possible to look at the many types of materials used in heat exchangers.

REFERENCES

1. M.D. Rajkamal, M. Mani Bharathi, Shams Hari Prasad, "Thermal Analysis of Shell and Tube Heat Exchanger", International Journal of Pure and Applied Mathematics, Volume 119 No. 12 2018, 14299-14306.
2. Shuvam Mohanty, Shofique Uddin Ahmed, "Performance prediction of Counter flow Heat Exchanger by using CFD technique", IJEDR, Volume 6, Issue 2, ISSN: 2321-9939.
3. Ankush S. Patil, H. S. Farkade, "Advances in Design and Development of Heat Exchangers: A Review", International Research Journal of Engineering and Technology (IRJET), Volume: 04, Issue: 05, May -2017.
4. Chandan Kumar Sethi, "CFD Analysis on Effectiveness of a Plate Type Heat Exchanger Using Sea Water and Engine Oil", International Journal of Advanced Mechanical Engineering, Volume 12, Number 1 (2017) pp. 191-198.
5. Deepa Shrivastava, Rahul Mishra, "CFD Analysis of Heat Transfer for Tube-In- Tube Heat Exchanger", International Journal of Recent Trends in Engineering & Research (IJRTER), Volume 03, Issue 08, August – 2017.
6. Dipankar De, Tarun K. Pal, Santanu Bandyopadhyay, "Helical baffle design in shell and tube type heat exchanger with CFD analysis", International Journal of Heat and Technology, Vol. 35, No. 2, June 2017, pp. 378-383.
7. Kvenumadhav, Sudhanshu Kumar, Chandrashekar Goud, "Performance Analysis of Different Heat Exchanger Design Using CFD Simulation", IJRAET, Volume 6, Issue 2 OCT 2017.
8. Mohammed Irshad, Mohammed Kaushar, G. Rajmohan, "Design and CFD Analysis of Shell and Tube Heat Exchanger", IJESC, Volume 7 Issue No.4, 2017.
9. Q. Miao, Z. Wang, L. Wang, "A Failure Analysis of Air Heat Exchanger Based on Modal Calculation and Thermal-Structural Interaction Simulation", International Journal of Materials, Mechanics and Manufacturing, Vol. 5, No. 3, August 2017.
10. Nakka Sita Rama Raju, V Jaya Prasad, S Kamaluddin, A Sunny Kumar, "Thermal Analysis of Shell and Tube Type Heat Exchanger to Demonstrate the Heat Transfer Capabilities of Various Thermal Materials Using ANSYS", International Journal of

- Mechanical Engineering and Technology (IJMET), Volume 8, Issue 2, February 2017, pp. 160–166.
11. R. Berlin Samuel Raj, "Heat Transfer Analysis of a Straight Tube and Helical Coil Heat Exchanger Using CFD", *Advances in Natural and Applied Sciences*. 11(6); Pages: 423-437, 2017.
 12. Santosh K Katarki, Anandkumar S Malipatil, "CFD Analysis of Shell and Tube Heat Exchanger for Heat Transfer Capabilities", *International Journal of Engineering and Techniques*, Volume 3 Issue 6, Dec 2017.
 13. Subin Michael, "Comparative CFD Analysis of Shell and Serpentine Tube Heat Exchanger", *International Research Journal of Engineering and Technology (IRJET)*, e-ISSN: 2395 -0056, Volume: 04 Issue: 02, Feb 2017.
 14. Basawaraj S. Hasu, J. Govardhan, Satyanarayana Rao, "Thermal Analysis of Tubular Heat Exchangers Using Ansys By Using Alloy Materials as Tube Materials", *Anveshana's International Journal of Research in Engineering and Applied Sciences*, Volume 1, Issue 3, 2016.
 15. K Ashok Reddy, "A Review of Heat Transfer Studies for Shell & Tube Heat Exchangers", *IJSDR*, Volume 1, Issue 5, May 2016.
 16. Pramod Deshmukh, Vikram D Patil, CFD Analysis of Heat Transfer in Helical Coil Tube in Tube Heat Exchanger by, *IJIERT*, 2016.
 17. J. Bala Bhaskara Rao, V. Ramachandra Raju, "Numerical and heat transfer analysis of shell and tube heat exchanger with circular and elliptical tubes", *Rao and Raju International Journal of Mechanical and Materials Engineering* Volume 11, Issue 6, DOI 10.1186/s40712-016-0059-x.
 18. D. Amrutha Vijay, P. Snehalatha, "Computer Aided Design and Analysis of a Heat Exchanger", *International Journal of Science, Engineering and Technology Research (IJSETR)*, Volume 5, Issue 7, July 2016.
 19. Digvendra singh, Narayan Das Pal, "Designing and Performance Evaluation of a Shell and Tube Heat Exchanger using Ansys", *International Journal of Scientific Engineering and Applied Science (IJSEAS) – Volume-2, Issue-3, March 2016*.
 20. Mayank Bhola, Vinod Kumar, Satyendra Singh, "Heat Transfer Enhancement in Concentric Tube Heat Exchanger in ANSYS FLUENT", *International Journal of Engineering Research & Technology (IJERT)*, Vol. 4 Issue 04, April-2015.
 21. Pankaj D. Lad, Santosh G. TAJI, Baban B. Londhe, "Optimization of Concentric Pipe Heat Exchanger as Per Asme Code using Finite Element Analysis", *International*

Journal on Recent Technologies in Mechanical and Electrical Engineering (IJRMEE),
Volume: 2 Issue: 10, ISSN: 2349-7947, 2015.

22. Roshan. V. Marode, Ashok. J.Keche, "Thermal Analysis Validation for Different Design Tubes in a Heat Exchanger", International Journal of Engineering Research and General Science Volume 3, Issue 1, January-February, 2015.
23. Saurabh Kumar, Neha Maheshwari, Brajesh Tripathi, "Computational Analysis of Different Nanofluids effect on Convective Heat Transfer Enhancement of Double Tube Helical Heat Exchanger", International Journal of Scientific Engineering and Applied Science (IJSEAS) - Volume-1, Issue-4, June 2015.
24. Sunil B. Revagade, Kalyani G. Deshmukh Gopal V. Shejole Vijay R, "Analysis of different type of tubes to optimize the efficiency of Heat Exchanger", IJSET - International Journal of Innovative Science, Engineering & Technology, Vol. 2 Issue 3, March 2015.