BRANDENBURGISCHE TECHNISCHE UNIVERSITÄT COTTBUS-SENFTENBERG

FACHGEBIET PROZESS- UND ANLAGENTECHNIK

MASTER THESIS

Topic: **Assessment and Design of a cooling system for explosive gases**

Presented by: Viral Rajubhai Mistry

Born on: 13.10.1995 in: Vadodara, India

to
achieve the academic degree

MASTER OF SCIENCE

(M.Sc.)

Brandenburg
University of Technology
Cottbus - Senftenberg

Supervisor: Prof. Dr.-Ing. Harvey Arellano-Garcia
First Reviewer: Dr.-Ing. Bogdan Dorneanu
Submission Date: 18.07.2023

BRANDENBURGISCHE TECHNISCHE UNIVERSITÄT COTTBUS-SENFTENBERG

FACHGEBIET PROZESS- UND ANLAGENTECHNIK

MASTER THESIS

Thema:	**Analyse und Entwurf eines Kühlsystems für Abgase**	
Präsentiert von:	Viral Rajubhai Mistry	
Geboren am:	13.10.1995	in: Vadodara, India

um den akademischen Grad zu erreichen

MASTER OF SCIENCE

(M.Sc.)

Brandenburg
University of Technology
Cottbus - Senftenberg

Vorsitzender:	Prof. Dr.-Ing. Harvey Arellano-Garcia
Gutachter:	Dr.-Ing. Bogdan Dorneanu
Einreichungsdatum:	18.07.2023

Part I.

Statement of Authorship

I hereby certify that I have authored this Master Thesis entitled ***Assessment and Design of a cooling system for explosive gases*** and the thoughts taken directly or indirectly from external sources (including electronic sources) are identified as such without exception. I am aware that violations of this declaration may lead to subsequent withdrawal of the degree.

Cottbus, 18.07.2023 Viral Rajubhai Mistry

Erklärung der Autorenschaft

Hiermit bestätige ich, dass ich diese Masterarbeit mit dem Titel ***Assessment and Design of a cooling system for explosive gases*** selbstständig und ohne unzulässige Unterstützung durch Dritte verfasst habe. Es wurden nur die in dieser Arbeit angegebenen Quellen und Hinweise verwendet. Mir ist bekannt, dass Verstöße gegen diese Erklärung zum nachträglichen Entzug des akademischen Grades führen können.

Cottbus, 18.07.2023 Viral Rajubhai Mistry

Acknowledgments

I would like to express my acknowledgement to few people who supported and kept me motivated to complete the master thesis successfully. To begin with, I would like to thank my parents and family members for their unconditional support and encouragement. It would not have been possible for me to reach this phase of my life without their love, sacrifices and blessings.

I would like to convey my sincere thanks Prof. Dr.-Ing. Harvey Arellano-Garcia (Chair of Process and Plant Technology) and Dr.-Ing. Bogdan Dorneanu, for providing continuous guidance throughout my entire thesis. The knowlegde and experience they have shared with me has helped me a lot in successfully completing the master thesis. Also, I am thankful for the amount of precious time they have invested in me.

I am extremly grateful to my supervisors M.Sc. Kai Risthaus and Dr. Ing. Dmitrij Laaber, for providing me the opportunity to do my research work as a master student at DLR (Deutsches Zentrum für Luft- und Raumfahrt). I am thankful for having shown the support and trust towards me. Both of them have guided me at each step towards my research and groomed me for being a better research engineer. The way they have conducted regular meetings for updates and brainstorming has really helped me in numerous ways. I really appreciate the fact that they have brought the best out of me as an engineer.

I am thankful to them to have motivated me multiple times when I needed support staying away from the family. Lastly, I would like to thank Ms. Dikshita Deka for her love and support that she had shown throughout my thesis.

Viral Rajubhai Mistry

Kurzfassung

Diese Arbeit befasst sich mit dem Entwurf und der Analyse eines Rohrbündelwärmetauschers zur Abkühlung der heißen Abgase. Darüber hinaus wurde eine vergleichende Studie zwischen dem klassischen und dem modifizierten Design des Wärmetauschers durchgeführt, um die Effektivität zu verbessern. Die Modifikation der Konstruktion umfasst schräge Umlenkbleche in einem Winkel von 30°. Diese Studie umfasst auch die Beobachtung des Strömungsmusters und die Veränderung der Art der Vermischung der Flüssigkeitsteilchen, die auch die Wärmeübertragung verändert.

Das Abgas ist eine Kombination aus Wasserstoff und Stickstoff mit einer Temperatur von 800 °C, während das Kühlmittel normales Leitungswasser mit einer Temperatur von 25 °C ist. Es wurden verschiedene Optionen für die Kühlung von Abgasen geprüft, und später wurde eine der besten verfügbaren Optionen in Form eines Rohrbündelwärmetauschers gewählt. Ein Rohrbündelwärmetauscher wird eingesetzt, da er einen effektiven Wärmeaustausch zwischen den Arbeitsmedien ermöglicht und die Möglichkeit einer weiteren Optimierung besteht. Außerdem ermöglicht der Rohrbündelwärmetauscher eine effiziente Kühlung durch die Kombination von Turbulenz und konvektiver Wärmeübertragung. Da der Wärmetauscher robust ist und im Vergleich zu anderen Typen eine bessere Effizienz gewährleistet, ist ein Rohrbündelwärmetauscher für diesen Prozess gut geeignet.

Mit Hilfe der numerischen Strömungsmechanik (CFD) umfasst die Forschung eine theoretische Untersuchung des Wärmeübertragungsprozesses sowie eine numerische Simulation der Fluidströmung und der Wärmeübertragung in beiden Systemen. Auf der Grundlage der Temperatur-, Druck- und Geschwindigkeitsdiagramme werden die Leistungen der beiden Konstruktionen bewertet. Es wurde versucht, die Effizienz des Rohrbündelwärmetauschers zu verbessern, ohne größere Änderungen an der Konstruktion vorzunehmen, und gleichzeitig wurden auch wirtschaftliche Überlegungen angestellt.

Aus den Ergebnissen geht hervor, dass der Einbau der schrägen Umlenkbleche einen besseren Wärmeübergangskoeffizienten und damit einen besseren thermischen Gesamtwirkungsgrad als bei der herkömmlichen Konstruktion ermöglicht.

Abstract

This thesis focuses on the design and analysis of a shell and tube heat exchanger for cooling down the hot exhaust gases. Additonally, a comparative study has been made between classical and modified design of the heat exchanger with an aim of improvement in effectivity. The modification of the design involves inclined baffles at an angle of 30°. This study also involves observing the flow pattern and the change in nature of fluid particle mixing which changes the heat transfer as well.

The exhaust gas is a combination of hydrogen and nitrogen at 800°C, while the cooling agent is normal tap water at 25°C. Different options for the cooling of exhaust gases have been assessed and later on, one of the best available options is chosen in the form of shell and tube heat exchanger. A shell and tube heat exchanger is utilized as it allows effective heat exchanger between the working fluids and there is a possibility of further optimization. Additionally, shell and tube heat exchanger enables efficient cooling through the combination of turbulence and convective heat transfer. As the heat exchanger is robust and ensures better efficiency compared to other types, a shell and tube heat exchanger is well suited for this process.

Using computational fluid dynamics (CFD), the research involves a theoretical investigation of the heat transfer process as well as a numerical simulation of the fluid flow and heat transfer in both systems. Based on the temperature, pressure and velocity plots, the performances of two designs are assessed. An effort has been made to improve the efficiency of the shell and tube heat exchanger without making major changes in the design and at the same time, economical considerations have also been made.

It is evident from the results that the incorporation of the inclined baffles facilitates better heat transfer coefficient and thus better overall thermal efficiency compared to the conventional design.

Contents

List of Figures

List of Tables

Acronyms

CFD	Computational Fluid Dynamics
PHE	Plate Heat Exchanger
HX	Heat Exchanger
LMTD	Logarithmic Mean Temoerature Difference
RANS	Reynolds Average Navier Stokes
LES	Large Eddy Simulation
RNG	Random Number Generator
DDT	Deflagration-to-Detonation Transition
ATEX	Explosive Atmosphere
CAD	Computer Aided Design
STEP	Standard for the Exchange of Data
IGES	Initial Graphics Exchange Specification
DOE	Design of Experiments

Symbols and Suffixes

T = Temperature [K]
Q = Heat duty [W]
F = Correction Factor
P = Effectiveness of the heat exchanger
R = Heat capacity ratio
U = Overall heat transfer co-efficient [W/m^2K]
r = Radius [m]
k = Thermal conductivity [W/mK]
h = Convective heat transfer co-efficient [W/m^2K]
A = Area $[m^2]$
N_t = Number of tubes
d = Tube diameter [m]
D_s = Shell diameter [m]
D_b = Bundle diameter [m]
D_c = Clearance diameter [m]
B_s = Baffle spacing [m]
N_b = Number of Baffles

1. Introduction

1.1. Background

In the recent times, hydrogen has proven to be a great alternative for the conventional fuels due to certain benefits. In order to obtain this hydrogen as a fuel, high temperature processes have to be done, which in turn release the exhaust gases at higher temperatures [1]. Additionally, the chemical industries have many experiments which have the exhaust gases at higher temperatures and sometimes these gases cannot be directly released into the environment due to certain safety reasons. They must be cooled down initially before releasing and if necessary, the gases must be made hazard free. There must be a cooling system which is capable of lowering the temperature of the gases.

A heat exchanger is used with the intention of exchanging thermal energy between two or more fluids which have a temperature difference. The primary function of a heat exchanger is either to cool down or heat up a fluid. This fluid might have single component or a mixture of compounds. The heat exchange occurs either with the mixing of fluids, or through a wall which separates the fluids [2]. Although, the mode of heat transfer subjects to the type of heat exchanger used. Heat exchangers are classified according to their method of functioning, flow arrangements, design and other features. Commonly their classification is based on:

- Type of heat transfer.
- Flow arrangements.
- Construction.
- Application, etc.

Heat exchangers used for cooling purposes are very crucial when it comes to process engineering. As the fluids are at elevated temperature, a risk of explosion and other safety hazards are always present. It is very essential to cool down the fluids. Selecting and designing of appropriate heat exchanger is necessary to ensure desired results. Thermal design of heat exchanger is an important step to begin with. It includes the calculation of various specifications of a heat exchanger, such as: the heat transfer area, dimensions of the heat exchanger unit, arrangements of the tubes, pressure drop etc [3]. This thesis mainly focuses on the development and simulation of a heat exchanger which is further used to cool down the flue gases. This work is carried out to reduce the potential risk of explosion due to high temperature of the flue gas.

A typical type of shell and tube heat exchanger is shown in the figure 1.1. As the name indicates, this particular heat exchanger consists of multiple tubes enclosed by a cylindrical shell where the heat exchange phenomenon takes place. The tubes are placed inside the shell in such a way that it offers maximum area for the heat exchange. The tube walls provide a separating boundary for the two fluids and offers the medium for heat transfer. Additionally, it ensures that the two fluids are not mixed with each other. These tubes are fixed with the tube sheet which acts as tube holding device and ensures the sealing to prevent the mixing of fluids [2]. This type of heat exchanger have other parts as well namely, baffles, nozzles, tie rods and many more, which would be discussed in upcoming chapter .

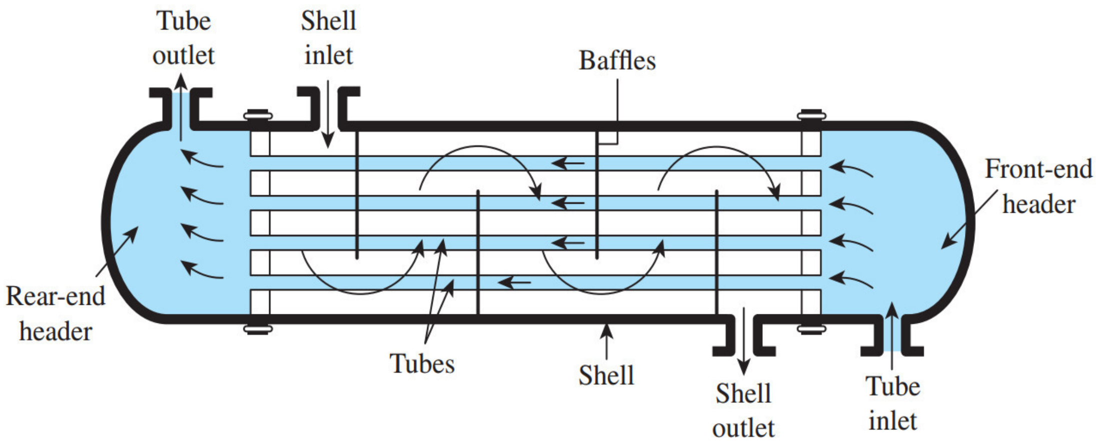


Figure 1.1.: Schematic diagram of a typical Shell and Tube Heat Exchanger [4]

Computational Fluid Dynamics (CFD) is a tool which helps in visualizing and computing the physical effects of the fluids. These physical effects for the motion of fluid are formulated using certain mathematical differential equations (namely Navier-Stokes equations). Solving these equations and simulating the fluid flow is carried out through CFD. It provides information and data about the flow properties which are otherwise difficult or not possible to study using the conventional methods. The CFD technique assists in optimizing the model and design of a component by considering the properties like the flow rate, heat transfer coefficient, temperatures of the respective fluids and many more.

1.2. Problem statement

Some of the processes in the industry leads to the exhaust gases leaving at very high temperatures. These flue gases should be cooled down up to certain temperature values before releasing them into the atmosphere or further reusing them as there is a risk of explosion due to the composition of flue gases. In this thesis the experiment with solar input at elevated temperature for creating fuel is considered for Deutsches Zentrum für Luft- und Raumfahrt, the experiment which is conducted involve reactive gases and probable fuels are also involved. As a consequence of this, safety hazard is inevitable.

In one of the high temperature elecrolyses experiments, the flue gas contains a binary mixture of Hydrogen and Nitrogen gas in different proportions and the temperature measured is 800°C. However, the self-ignition temperature of hydrogen gas is 585°C. Therefore, it is necessary to cool down the gas below its self-ignition temperature before releasing it to the ambience. This is a common problem for different experiments but currently, there is no general solution. This leads to the requirement of a standard cooling system.

1.3. Scope

The portrayed thesis work is focussed on the design and analysis of a cooling system for the flue gases, in the form of shell and tube heat exchanger, to cool them down below their self ignition temperature and hence mitigating the explosion hazard.

Chapter 2 discuses the state of the art of heat exchangers, their classification, materials used and the analytical method.

Chapter 3 gives insight about the Computational Fluid Dynamics (CFD) simulations for the heat exchangers.

Chapter 4 provides a short summary for the workflow of the study and the Computational Fluid Dynamics (CFD) process.

Chapter 5 provides an idea about the design and simulation results of the heat exchanger.

Chapter 6 discusses the results of the CFD simulation and a brief discussion for the same.

Chapter 7 concludes the thesis and also gives the idea for possible future work.

2. State of the art

In this chapter the functioning, classification and the materials of the heat exchangers will be briefly discussed. A detailed focus can be seen on shell and tube heat exchangers. Additionally, this chapter is about the designing criterion for the shell and tube heat exchangers.

The heat exchangers are primarily used to transfer the thermal energy which aims on cooling or heating a stream of fluid. A various types of heat exchangers are present which have different principles of operation and they differ in their construction as well.

2.1. Heat exchanger types

Heat exchangers are the devices with multiple purposes, e.g. energy saving, safety requirements, improving the efficiency of a plant and many more. Heat exchangers are classified as per the construction geometry, method of heat transfer, number of passes and the flow arrangements to name a few. Although, they are mainly classified according to the nature of heat transfer, that is, direct and indirect type of heat exchangers. A brief explanation of heat exchangers classified as per the construction is done here.

2.1.1. Double pipe heat exchanger

A double pipe heat exchanger, as shown in fig 2.1, consists of two concentric pipes having different diameters. The double pipe heat exchangers are quite bulky and sometimes are not cost effective. The primary application of the double pipe heat exchangers is for the process fluids which require smaller heat transfer areas (50-60 m^2)

In order to cater the requirements for mean temperature difference and the required values for pressure drops, the heat exchangers can be connected in parallel or series [6]. The pipes can also be bend in U shape to increase the amount of heat transfer if there is a limitation of space.

The double pipe heat exchangers are also termed as hairpin heat exchangers and can be used under fouling situations as well. The main reason for the above mentioned condition is the ease of maintenance and cleaning.

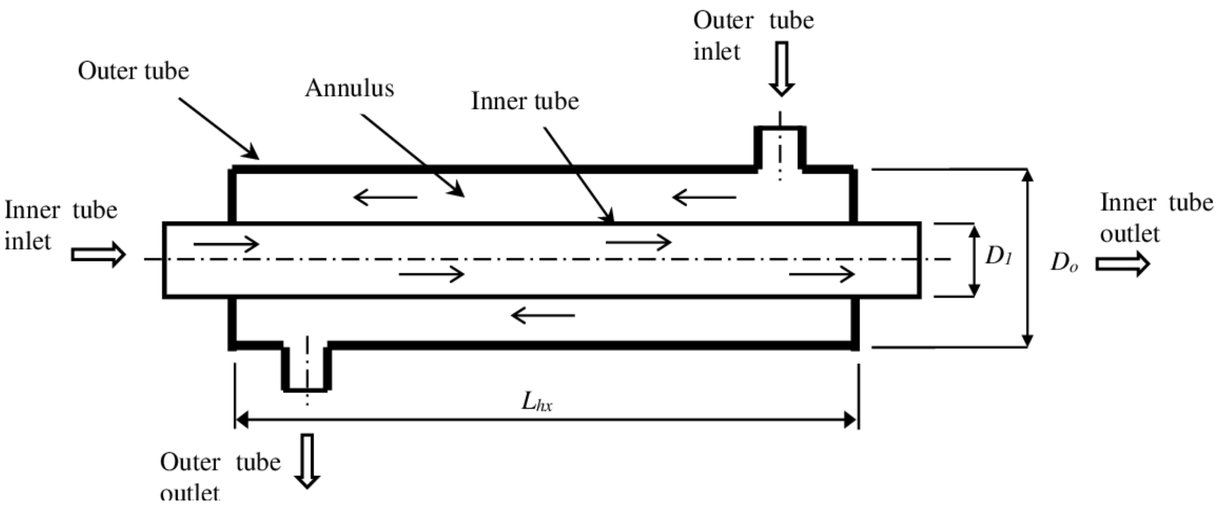


Figure 2.1.: Double pipe heat exchanger [5]

2.1.1.1. Advantages:

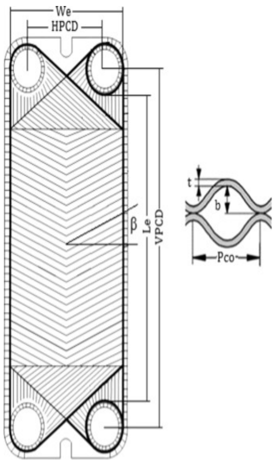
- Smaller in size.
- Simpler design.
- Not much space is required for maintenance.
- Standardized parts, etc.

2.1.2. Plate heat exchangers

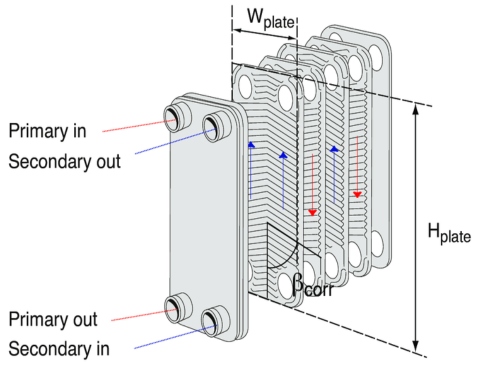
The plate heat exchangers (PHEs), as the name states, use thin plates as a medium of heat transfer as they form the flow channels. The plate surfaces may be smooth or they have corrugated fins which separate the fluids. The corrugations provided on the plates improve the heat transfer. A number of plates are connected using bars and frames. The plates have an alternating gasket pattern. These two plates are connected alternatively in equal or unequal numbers to form a series of plates. The parameters such as flow velocities, required heat transfer and pressure drops determines the arrangement of corrugation [6].

2.1.2.1. Extended surface heat exchangers

The heat exchangers of this type have fins provided on their primary heat transfer area. The fins are provided with an inention to increase the area of heat transfer. The primary application of the extended surface heat exchangers is for gas-to-gas and gas-to-liquid heat transfer [6]. The extension of the heat transfer is done on the gas side of the heat exchanger due to lower heat transfer coefficient of gases compared to liquid.



(a) Geometric representation of a single plate [7]



(b) Assembly of plates [8]

Figure 2.2.: Plate heat exchanger (PHE)

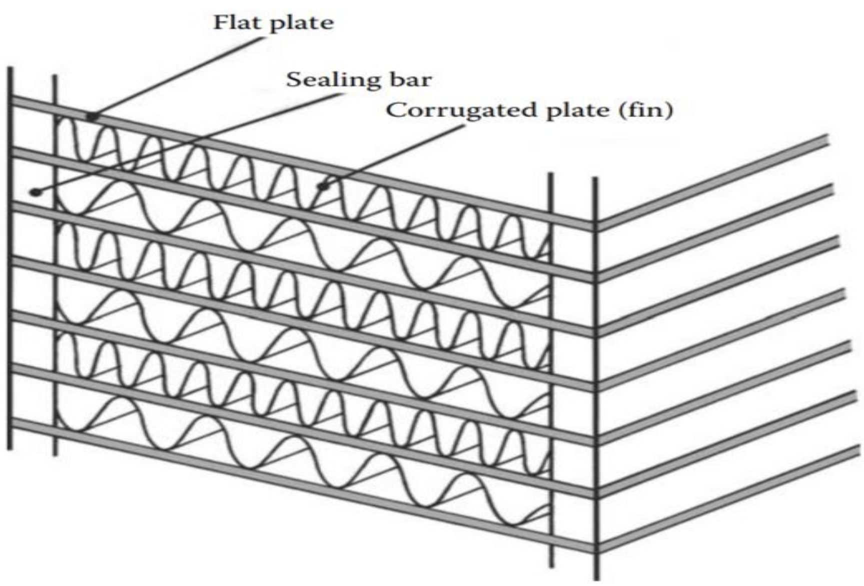


Figure 2.3.: Extended surface (Plate fin) type heat exchanger [6]

2.1.3. Shell and tube heat exchanger

As the name indicates, shell and tube heat exchanger contains a number of tubes enclosed in a shell. One of the fluids flows inside the tube while the other fluid flows across the tubes within the cells. The tube walls provide a solid boundary separating the two fluids [9]. Depending on the required heat transfer area, different arrangements are possible. Provision of baffles is an effective solution to improve the heat transfer by allowing the fluid inside the shell to flow in a zig-zag pattern [10].

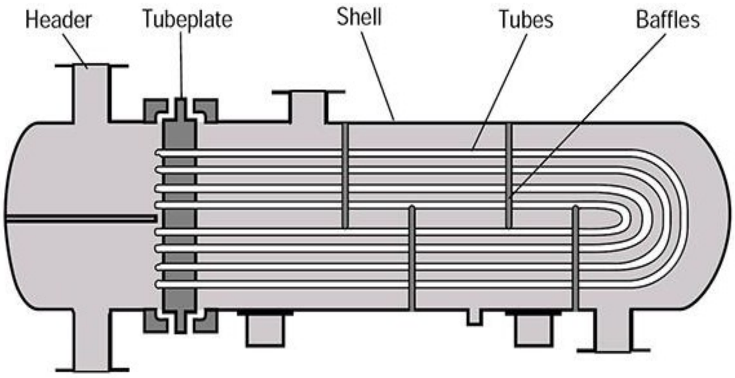


Figure 2.4.: U-tube type shell and tube heat exchanger [11]

The shell and tube heat exchangers can be further mainly classified into three types, namely fixed tubesheet design, the U-tube design and floating head type. A common trait in these three types is the stationary front-end head while the rear-end head might be floating or stationary. The design of the rear-end head depends on the pressure and the thermal stresses [2].

2.1.3.1. Fixed tubesheet heat exchanger

In this design, the tubesheets that support the tube bundles, as shown in figure 2.5, are fixed to the shell by welding. This creates a fixed joint which is inflexible so it does not allow the stresses to settle, which are caused due to temperature difference [12]. This phenomenon leads to the requirement of special arrangements such as expansion joints. The main benefit of using fixed tubesheet heat exchanger it the low cost due to the simple construction. As the tube bundle is fixed to the shell and cannot be removed, it is difficult to clean the tubes from the outer side. Thus, the usage of fixed tubesheet heat exchanger is particularly limited to certain shell side fluids which have cleaner services [13].

2.1.3.2. U-tube heat exchanger

As seen in figure 2.4, the U-tube heat exchangers have only one tubesheet and it supports both the ends of a tube. The tube is bent in a U shape and it serves as two tubes. U-tube heat exchangers are easy and comparatively inexpensive to manufacture [12]. As the other end of the heat exchanger is free, it allows the space for the tubes to expand and contract according to the stresses. Additionally, as the tube bundle is removable, the cleaning of the tubes from outer side is possible. Although, the inner surfaces of the tubes cannot be cleaned effectively at the U-bends [15]. It requires special cleaning devices such as flexible shaft cleaners. It is advisable to use the U-tube heat exchangers for the cleaner tube side fluids.

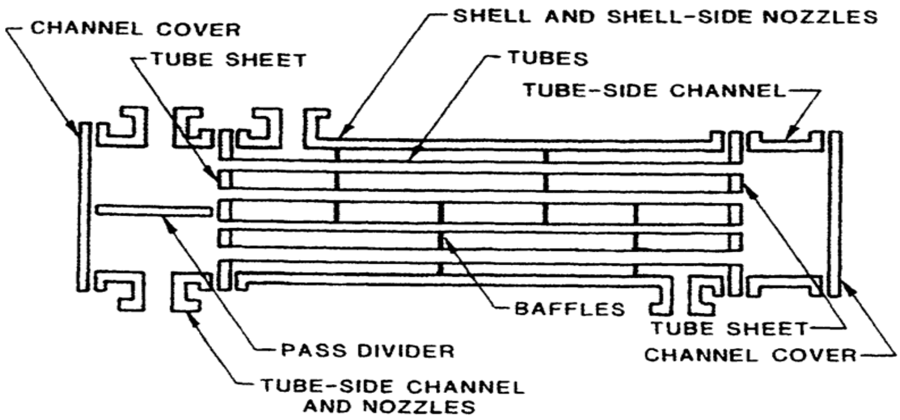


Figure 2.5.: Fixed tube type shell and tube heat exchanger [14]

2.1.3.3. Floating head heat exchanger

As the name suggests, the rear end head is floating and it allows free expansion of the tube bundle. These type of heat exchangers are the most flexible ones but the cost increases evidently. The tubes are straight and can be removed easily so it allows cleaning of tube from both the sides. Thus, it enables the floating head heat exchanger to be used for applications where both the fluid are dirty or have a tendency to degrade the tubes [15].

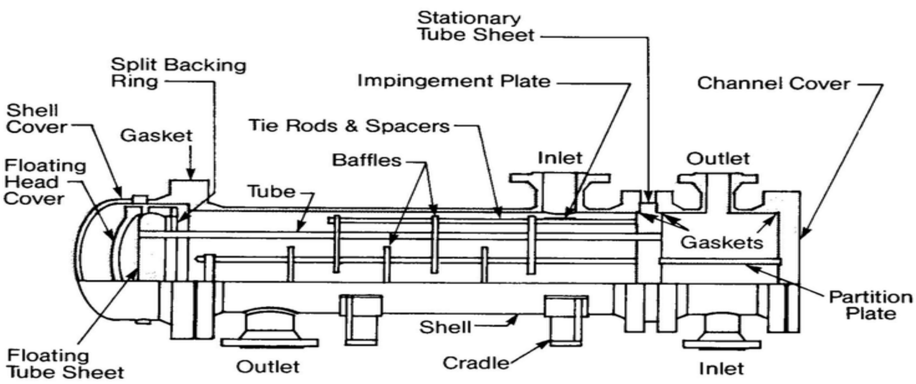


Figure 2.6.: Floating head type heat exchanger [16]

A comparison of the above mentioned configurations has been done in the table 2.1.

2.2. Heat exchanger design considerations

The shell and tube heat exchanger has many parts and accessories for it's proper functioning and efficiency.

Table 2.1.: Comparison of different configurations of the heat exchanger

Design characteristics	Fixed tube sheet	U-tube	Floating head
Thermal expansion	Expansion joint in shell	Tubes allow expansion	Floating head allow expansion
Bundle removal and replacement	No	Yes	Yes
Tube replacement	Yes	Limited to outside row	Yes
Mehanical internal tube cleaning	Yes	Yes, specific tools required	Yes
Operating temperature	Low	High	Moderate

Some of the important parts which reqiure proper designing as per thermal calculations are shell, tube bundle, baffles, tube sheet, head cover, tie rods and spacers. The thermal design is done in regards with the estimation of parameters such as the required heat transfer area, the geometrical dimensions of the shell and number of tubes required, the tube pitch, baffles and baffle spacing, the number of passes and the pressure drop on individual sides.

2.2.1. Shell

The shell is a cylindrical vessel of the heat exchanger which encloses the tube bundle. The manufacturing of shell is generally done by rolling and welding sheets of stainless steel into pipes. The diameter of the shell depends on the number of tubes in the tube bundle and a certain amount of clearance must be provided as well [17]. The clearance is determined according to the type of heat exchanger. The diameter of the shell should be chosen to provide a perfectly tight fit with the tube bundle.

2.2.2. Tube and tube pitch

Tubes carry one of the fluids inside the heat exchanger. The number of tubes are calculated using the knowledge of the required heat transfer area and the area of one tube. Selecting the approriate dimensions of the tube is very important. If the tubes are longer, the shell diameter is reduced and an increment can be observed in the shell side pressure drop [17]. Most common materials used for the tubes include stainless steel, copper-nickel alloy and bronze. The selection of these materials depend on the fluid and the operating temperature. If there is a necessity to increase the heat transfer coefficient, additional projections in the form of fins can be provided on the outer surface of the tubes.

The minimum distance between the centres of two consecutive tubes is defined as the tube pitch. The tube pitch determines the size of tube bundle and ultimately the shell diameter [12]. The arrangement of the tubes is done either in triangular or square pattern, this arrangement of the tubes is termed as the tube layout.

Table 2.2.: Comparison of different tube layouts [3]

Arrangement	Triangular pitch	Square pitch
Turbulence	Higher	Lower
Heat transfer coefficient	Higher	Lower
Pressure drop	Higher	Lower
Heat transfer area	Lower	Higher
Number of tubes	Lower	Higher

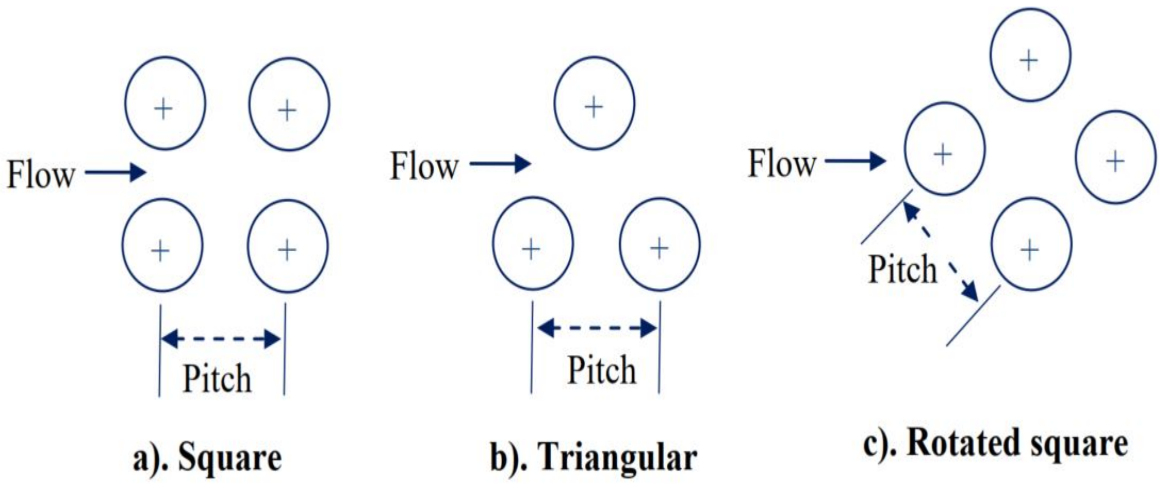


Figure 2.7.: Tube layout types [17]

2.2.3. Tube sheet

Tube sheet is a structure which supports the tube bundle and acts as a separator of the two fluids. The tubes are fixed on the tube sheet with some packing material or in some cases by welding. The thickness of tube sheet depends on the tube outer diameter, it should not be less than the outer diameter of the tubes [17].

2.2.4. Baffles

The main idea behind providing the baffles is to increase the heat transfer co-efficient. Baffles increase the fluid velocity by varying the cross sectional area and increasing the turbulence. The baffles maybe of a disc shape, cut-segmental shape or disc and doughnut shape [12].

The number of baffles are determined according to the length of the tubes. The distance between two consecutive baffles is termed as baffle-spacing.

2.2.5. Allocation of fluids

The allocation of the fluids on either side (shell side or the tube side) of the heat exchanger is important. Several fluid properties, such as viscosity, pressure, temperature, corrosive tendency, flow rate and fouling tendency, determine the allocation side in the heat exchanger [18]. Selecting a proper side for the fluids helps in designing an efficient and low cost manufacturing of the heat exchanger. To give an example, the fluids which are corrosive in nature and have a tendency to cause fouling are generally allocated to the tube side as the tubes are easy and inexpensive to replace [17]. In the table 2.3, the fluid allocation suggestions according to the properties is provided.

Table 2.3.: Fluid allocation to the respective sides

Tube side	Shell side
Corrosive, fouling fluid	Fluid with higher flow rate
Less viscous fluid	More viscous fluid
Hotter fluid	High temperature change fluid
Toxic or dirty fluid	Clean fluid

Sometimes, a trade-off shall be made while selecting the fluid for the respective sides considering the heat transfer co-efficient, pressure drop and economic aspect in mind. So, the above mention table is not striclty to be followed but some ammendments are allowed .

2.2.6. Heat exchanger design procedure

The shell and tube heat exchanger must be designed in such a way that it is capable of providing certain amount of pressure drop. To begin the design, all the data (such as flow rates, the composition of fluids, the pressure and temperature values etc.) which is required to design the heat exchanger should be made available so that the basic assumptions about the design data can be made [19]. Using the assumptions, the basic configuration (Fixed tube, U-tube or floating head) of the heat exchanger can be chosen. The second step in the design procedure is to estimate the heat exchanger dimensions, including the tube dimensions as well. This design is to be further verified and rated, if it is in the allowable limits, the design is confirmed else a new iteration of the design has to be carried out. The heat exchanger is designed using an iterative method which co-relates the data by simple equations is defined by Kern and termed as the Kern method [19].

2.2.6.1. Calculation of the energy balance

Once the required data is available, i.e. the thermophysical properties of both the fluids, the detailed calculation has to be done. This step is done to carry out the energy balance and furthermore, calculating the heat duty (Q) for the heat exchanger [19]. The equation 2.1 is used to calculate the value of heat duty. For the calculation of Q, three values of temperature must be known and the remaining temperature value is calculated using the knowledge of mass flow rates.

$$Q = m_c \cdot c_{pc} \cdot (T_{co} - T_{ci}) = m_h \cdot c_{ph} \cdot (T_{hi} - T_{ho}) \tag{2.1}$$

Here,

Q = required heat duty $[W]$

m = mass flow rate $[kg/s]$

T = Temperature $[K]$

the suffix c is used for cold fluid and h for the hot fluid while the suffixes i and o represents the inlet and outlet respectively.

2.2.6.2. Logarithmic mean temperature difference

As the name indicates, logarithmic mean temperature difference (LMTD or ΔT_{lm}) is the logarithmic average of the difference of temperatures of both the fluid streams at the particular ends of heat exchanger [18]. This temperature difference is used to design the heat exchanger by calculating the driving force for the heat transfer process. ΔT_{lm} for counterflow heat exchanger is calculated using equation 2.2

$$\Delta T_{lm} = \frac{(T_{hi} - T_{co}) - (T_{ho} - T_{ci})}{\ln\left(\dfrac{T_{hi} - T_{co}}{T_{ho} - T_{ci}}\right)} \tag{2.2}$$

The heat transfer largely depends on the ΔT_{lm}. The greater the value of ΔT_{lm}, the better is the heat transfer.

2.2.6.3. Correction factor

A correction factor (F) for the calculated ΔT_{lm} is to be determined and it is done using the thermal relation chart for different configurations of the heat exchanger. Generally, the correction factor has to be applied in the case of multipass configurations. For a shell which has the number of passes in the multiples of 2, the value of correction factor F, lies between 0.8 and 1 [20]. This F is then further multiplied to ΔT_{lm} and the corrected $\Delta T_{lm}\prime$ is further used in estimating the area of the heat exchanger. The equations 2.3 and 2.4 are used to calculate the parameters P and R respectively.

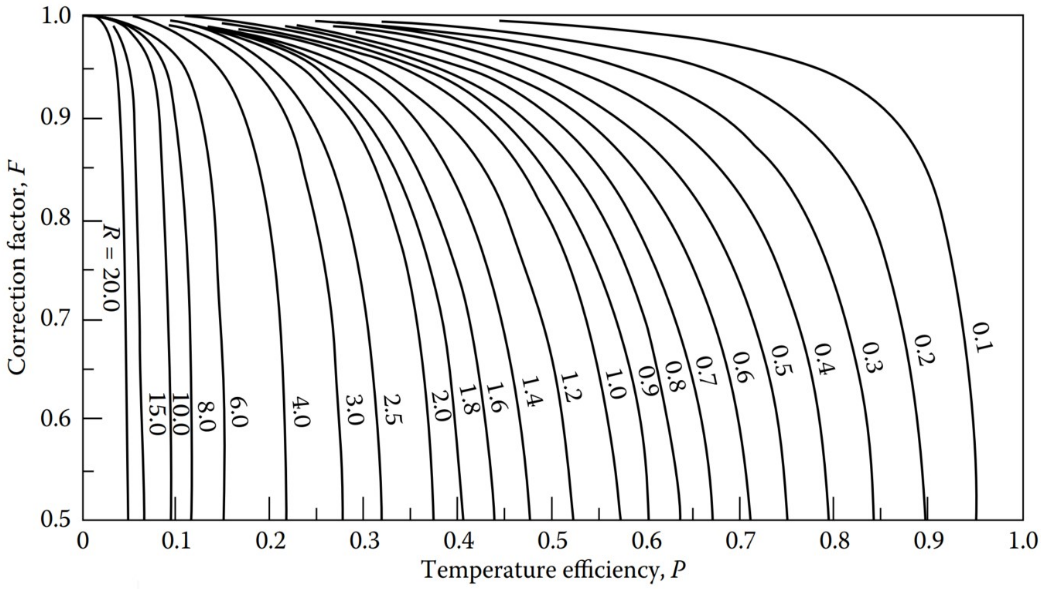


Figure 2.8.: LMTD correction factor [6]

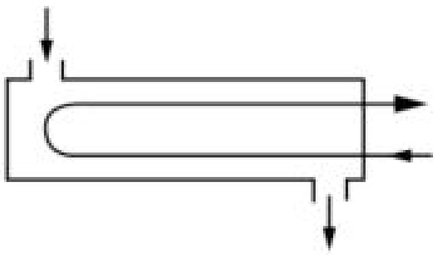


Figure 2.9.: Representation for P and R calculation [6]

$$P = \frac{T_{ho} - T_{ci}}{T_{hi} - T_{ci}} \tag{2.3}$$

$$R = \frac{T_{hi} - T_{ho}}{T_{co} - T_{ci}} \tag{2.4}$$

The term P is the effectiveness of the heat exchanger, calculated by dividing the actual heat transfer with the heat transfer if the temperature of the stream containing colder fluid is risen to match with the inlet temperature of hot fluid stream [20]. While, the term R is the heat capacity ratio which is obtained as a ratio of difference of temperature of the hot fluid to the cold fluid. Using the values of P and R, the value of F is extrapolated on the graph shown in the figure 2.8. It shows the correction factor chart with the X-axis showing the temperature efficiency or the temperature effectiveness and on the Y-axis, it measures the correction factor.

2.2.6.4. Overall heat transfer co-efficient

The overall heat transfer co-efficient (U) is the co-efficent or proportionality factor which determines the intensity of the heat transfer. It is the measure of the sum of individual thermal resistances including the thermal resistance provided by a wall. The overall heat transfer co-efficient is important for the calculations while designing any cooling or heating equipments. While designing the heat exchanger, the overall heat transfer co-efficient is the sum of the the thermal resistance provided by the wall, inner side of tube and outer side by the shell [6]. The equation 2.5 accounts for the overall heat transfer co-efficient (U) for the shell and tube heat exchanger.

$$U = \frac{1}{\frac{1}{r_i \cdot h_i} + \frac{\ln \frac{r_o}{r_i}}{k} + \frac{1}{r_o \cdot h_o}} \tag{2.5}$$

Here,

U = Overall heat transfer co-efficient $[W/m^2K]$

r = Radius $[m]$

k = Thermal conductivity $[W/mK]$

h = Convection heat transfer co-efficient $[W/m^2K]$

2.2.6.5. Calculation for the heat transfer area

The required heat transfer area is then calculated using the equation 2.6. This equation uses the previously calculated values of Q, ΔT_{lm}, F and U to find the required heat transfer area.

$$A = \frac{Q}{U \cdot \Delta T_{lm} \cdot F} \tag{2.6}$$

Here,

A = Area $[m^2]$

Q = Heat duty $[W]$

U = Overall heat transfer co-efficient $[W/m^2K]$

ΔT_{lm} = Logarithmic mean temperature difference $[K]$

F = Fouling factor

2.2.6.6. Tube dimensions and tube count

This is the step where the first assumption is to be made. Generally, there are a lot of options available for the selection of tube diameters and the thickness but according to the German standard for the tubes that are being used in shell and tube heat exchangers (DIN 28180 [12]) have their outer diameters in the range of 16, 20, 25, 30 and 38 mm. Addtionally, the most favourable option for the use in process engineering industries is the tube with OD 25 mm and 2 mm of wall thickness [12].

For transfering the heat efficiently, a sufficiently large heat transfer area is required, which is not fulfilled by a single tube. The above mentioned reason leads to the use of multiple tubes and an appropriate number of tube has to be calculated to attain the desired amount of heat transfer. The tube count N_t, is estimated using a ratio of the shell area to the area of a single tube is calculated. The equation 2.7 give the number of tubes.

$$N_t = \frac{A_s}{A_t} \tag{2.7}$$

Here,

A_s = Area of shell $[m^2]$

A_t = Area of a single tube $[m^2]$

The tube pitch p_t determines the size of tube bundle and ultimately the shell diameter. Additionally, the pitch also has a influence on the velocity of the fluid and pressure drop on the shell side. For the square pitch, the equation 2.8 gives the recommended tube pitch.

$$p_t = 1.25 \cdot d_o \tag{2.8}$$

Here,

d_o = outer diameter of the tube $[m]$

Once the tube pitch is known, it helps in determining the size of the tube bundle which greatly influences the size of the shell. For the calculation of the tube bundle diameter D_b, the tube outer diameter, the tube count, and two constants K_1 and n_1 are required. As the name states, the bundle diameter D_b is defined as the diameter which accomodates the bundle of all tubes into the shell. The bundle diameter is calculated using the equation 2.9.

$$D_b = d_o \cdot \left(\frac{N_t}{K_1}\right)^{\frac{1}{n_1}} \tag{2.9}$$

The constants K_1 and n_1 are mentioned in the table 2.4 and the constants must be selected as per the configuration of the heat exchanger (No. of passes). This estimation of the D_b, leads to the calculation for the shell diameter with the help of the clearance.

Table 2.4.: Constants for tube bundle [6]

No. of passes	1	2	4	6	8
K_1	0.215	0.156	0.158	0.0402	0.0331
n_1	2.207	2.291	2.263	2.617	2.643

2.2.6.7. Calculation for the shell diameter

The calculation of the shell diameter (D_s) comprises of two variables, namely the bundle diameter (D_b) and the bundle clearance diameter (D_c). The equation 2.10 shows the calculation for the shell diameter.

$$D_s = D_b + D_c \tag{2.10}$$

The clearance diameter is obtained by extrapolating the bundle diameter with the type of the cylinder head as per the application in the figure 2.10.

2.2.6.8. Calculation for the baffles

Baffles are one of the most important parts of the heat exchanger when it comes to increasing the heat transfer co-efficient. They work by icreasing the velocity of the fluid and by increasing the turbulence. To calculate the number of baffles N_b, initially, the baffle spacing B_s has to be calculated. Equations 2.11 and 2.12 are used to calculate B_s and N_b respectively.

$$B_s = 0.4 \cdot D_s \tag{2.11}$$

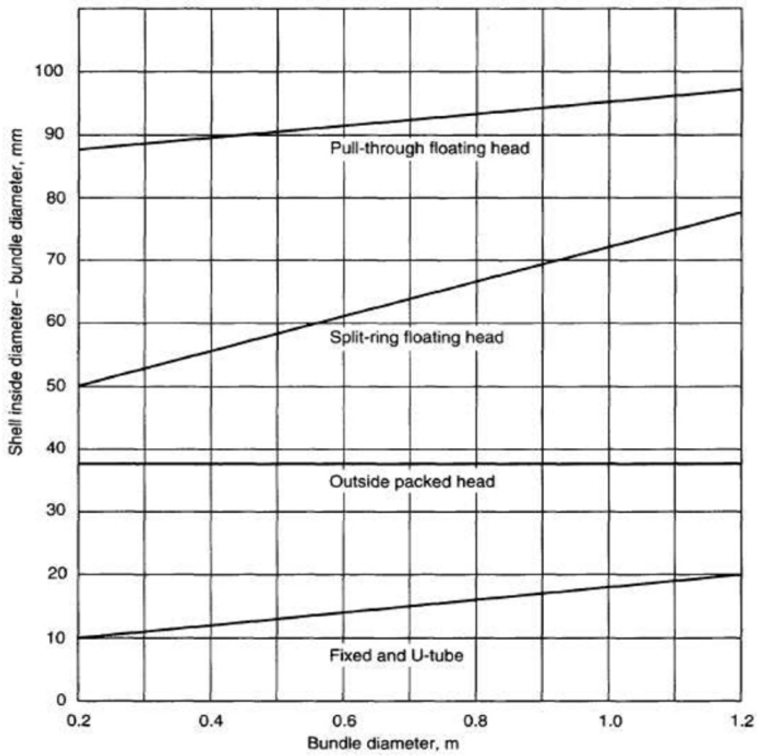


Figure 2.10.: Bundle clearance diameter [20]

$$N_b = \frac{Length\ of\ tube}{B_s} \tag{2.12}$$

Once these steps are completed, the next part according to the Kern method is the calculation of the individual heat transfer co-efficient (shell side and tube side heat transfer co-efficient) and comparing with the previously calculated overall heat transfer co-efficient. Later, the pressure drop is determined and if the pressure drop is in allowable limits, the design is acceptable for the CFD or verification. Else, the new estimation for the tube dimension has to be done and the whole procedure has to be carried out once again.

3. Computational Fluid Dynamics for heat exchanger models

Computational fluid dynamics (CFD) is the field of mechanics which is related to the simulation of the numerical flow that has the capabillity of solving complex problems. A system has to be developed, using the equations for conservation (mass, momemtum and energy), continuity and the impulse of the fluid known as the Navier-Stokes equations, for solving the problems [21]. The study for the flow patterns which might be difficult using the conventional methods, can be done with the help of CFD. For modelling the behaviour of any component, certain steps must be followed with utmost care. There is a possibility of error and sometimes faliure, if there is a negligence while modelling or discretizing. In the recent times, a lot of CFD simulation tools are available commercially which predict the flow with high resolution in space and time and for a number of desired quantities.

CFD has numerous applications, such as aersopace industry, automotive industry, chemical industry, defense sector, for hazard and safety analysis to name a few. However, the accuracy of the CFD simulation results is never 100%. There is a number of reasons for the inaccuracy, which includes uncertainity in the input data, handling error, limited availability of the resources and others. These inaccuracies can be improved with experience, knowledge or validation experiments.

CFD for the heat exchangers is helpful in the iterative sizing as well as in the rating of a heat exchanger by the design of experiments and helps in saving time and investments [22]. The heat exchanger problem for a CFD simulation tool is complex so when it comes to the calculation part, large computer memory and the computing power is need which consumes a lot of time [23]. This problem can be tackled using a distributed resistance approach; which means the shell side of the heat exchanger can be meshed using comparatively coarser grids [24].

3.1. Governing equations

CFD uses certain governing equations and assumptions at the same time. To solve the CFD problem using a tool, the equations of continuity and momentum must be solved and also the equations of heat transfer. For this thesis, ANSYS Fluent 2022 R1 is the CFD simulation tool which is being used to run the calculations.

The below listed governing equations are solved depending on the nature of the flow and the turbulence model in Ansys Fluent and the results in the form of contours, plots or graph are presented.

3.1.1. The mass conservation equation

According to the mass conservation equation, for a closed system, the quantity of mass can neither be removed nor added [25]. The equation 3.1 is the mass conservation equation mentioned in the Fluent Theory guide [26].

$$\frac{\partial /rho}{\partial t} + \nabla(\rho^* V) = 0 \tag{3.1}$$

Here, ∇ represents the Nabla operator, ρ is the density, t id the time and V is the velocity of the flow [25].

3.1.2. The momentum equation

The law of conservation of momentum, as known, is based on the "Newton's second law of motion" and is defined as the product of mass and acceleration of a fluid element and is equivalent to the summation of forces acting on it. The equation 3.2 is used for the conservation of momentum within an inertial frame of reference [26].

$$\frac{\partial}{\partial t}(\rho \vec{v}) + \nabla(\rho \vec{v} \times \vec{v}) = -\nabla p + \nabla(\bar{\bar{\tau}}) + \rho \vec{g} + \vec{F} \tag{3.2}$$

Here, F is the net external body force, ∇p represents the pressure gradient, $\rho \vec{g}$ is the net gravitational force, and $\bar{\bar{\tau}}$ is the stress tensor which is the representation of the equation 3.3.

$$\bar{\bar{\tau}} = \mu \left[(\nabla \vec{v} + \nabla \vec{v}^T) - \frac{2}{3} \nabla \vec{v} I \right] \tag{3.3}$$

Where, μ is used for the molecular viscosity and the unit tensor is represented by I.

3.2. Turbulence modelling

Turbulence is defined as an irregular or chaotic fluid motion due to high velocity and randomness of the move. The instability of the flow causes turbulence and it can be identified with the knowledge of Reynolds number [27]. At high Reynolds number, the dominance of viscous forces over the inertial forces is lost and thus the system to keep the fluid motion in laminar path. This is the main cause for turbulence at high Reynolds number.

There are a number of turbulence models classified as per the degree of resolution and all of them have different type of approximations to solve the Navier-Stokes equation. While talking about the CFD simulation tools, there are several turbulence models developed based on certain criterions, namely the Reynolds Average Navier-Stokes (RANS) model and the Large Eddy Simulation (LES) model [27]. The RANS model is characterized according to the flow properties, i.e. the standard $k-\epsilon$ model, realizable $k-\epsilon$ model, $k-\epsilon$ RNG (random number generator) model and the $k-\omega$ model.

3.2.1. The RANS model

The RANS model has an assumption of that the turbulence can be simply modelled using the transport equations. In order to model the turbulence using the RANS model, a new term has to be added in the Navier-Stokes equation, termed as the Reynolds Stress tensor [28].

The RANS model uses the phenomenon of fluctuation of the flow characteristics such as temperature, pressure, velocity and density (for compressible flows). Taking the velocity component into account, decomposing it into an average value and a fluctuation component [28], the turbulence fluctuation is obtained as

$$u = \bar{u} + u' \tag{3.4}$$

In the above equation, u' is the fluctuating velocity and $\bar{u}$ is the average velocity component [29]. The average veloctiy is expressed as

$$\bar{u}(x) = \lim_{\Delta t \to \infty} \frac{1}{\Delta t} \int_{t_1}^{t_1+\Delta t} u(x,t)dt \tag{3.5}$$

The pressure component is similarly modelled and the decompostion of both the pressure and veloctiy components represented in Navier-Stokes equation is obtained as equation 3.6.

$$\frac{\partial \langle U_i \rangle}{\partial t} + \langle U_i \rangle \frac{\partial \langle U_i \rangle}{\partial x_j} = -\frac{1}{\rho}\frac{\partial}{\partial x_j}\left\{ \langle P \rangle \delta_{ij} + \mu \left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) - \rho \langle u_i u_j \rangle \right\} \tag{3.6}$$

Here, $-\rho\langle u_i u_j \rangle$ represents the Reynolds stresses induced for the variation of the velocity profile due to high Reynolds number. To finalize the equation, this term has to be modelled. The Boussinesq approximation states that the Reynolds stresses are in direct proportion to the mean velocity gradient [29]. By applying the Boussinesq approximation, the above equation finally becomes,

$$\frac{\partial\langle U_i\rangle}{\partial t} + \langle U_i\rangle\frac{\partial\langle U_i\rangle}{\partial x_j} = -\frac{1}{\rho}\frac{\partial\langle P\rangle}{\partial x_i} - \frac{2}{3}\frac{\partial k}{\partial x_j} + \frac{\partial}{\partial x_j}\left[(v+v_t)\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right)\right] \tag{3.7}$$

To use the RANS model, extra transport equations are required, the first equation accounts for the turbulent kinetic energy k while the later for turbulent dissipation ratio ϵ or the specific dissipation rate ω.

3.2.1.1. The two-equations model

As the name implies, the turbulence model that needs two additional equations to define it completely. The classification is done according to the number of extra equations that are required. The zero equation model assumes the viscosity to be constant so no extra equation is needed while the one equation model makes the assumption viscosity somehow affects the turbulence so it needs an extra equation. For the two equations model, the extra equations are required for the turbulent velocity. There are certain two equation models which can be applied for turbulence modelling and the models are stated below.

$k-\epsilon$ **model :** Kinetic energy (k) being the first variable and the turbulent dissipation (ϵ) being the second [30], which are transported, the transport equations for the respective variables are:

$$\frac{\partial k}{\partial t} + \langle U_j\rangle\frac{\partial k}{\partial x_j} = \nu_T\left[\left(\frac{\partial\langle U_i\rangle}{\partial x_j} + \frac{\partial\langle U_j\rangle}{\partial x_i}\right)\frac{\partial\langle U_i\rangle}{\partial x_j}\right] - \varepsilon + \frac{\partial}{\partial x_j}\left[\left(\nu + \frac{\nu_T}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] \tag{3.8}$$

$$\frac{\partial\varepsilon}{\partial t} + \langle U_j\rangle\frac{\partial\varepsilon}{\partial x_j} = C_{\varepsilon 1}\nu_T\frac{\varepsilon}{k}\left[\left(\frac{\partial\langle U_i\rangle}{\partial x_j} + \frac{\partial\langle U_j\rangle}{\partial x_i}\right)\frac{\partial\langle U_i\rangle}{\partial x_j}\right] + C_{\varepsilon 2}\frac{\varepsilon^2}{k} + \frac{\partial}{\partial x_j}\left[\left(\nu + \frac{\nu_T}{\sigma_\varepsilon}\right)\frac{\partial\varepsilon}{\partial x_j}\right] \tag{3.9}$$

The results obtained from the $k-\epsilon$ model are not always reliable as the model cannot accurately predict the streamline curvature or swirling flows. Additionally, in this model, the parameters are a compromise for achieving best performances for a variety of flows. Due to the above mentioned points, a different model is derived using the $k-\epsilon$ model and it is known as realizable $k-\epsilon$ model. In the above mentioned equations, $C_{\varepsilon 1}$, $C_{\varepsilon 2}$, are the closure coefficients.

Realizable $k-\epsilon$ model : In the standard $k-\epsilon$ model, the normal stress might become negative. The realizable $k-\epsilon$ model is a correction of the standard $k-\epsilon$ model where the realizability constraint is provided. The prime difference of this model is, here C_μ is not a constant and the dissipation rate has a new transport equation [30].

For the normal component of the stress tensor,

$$\langle u_i u_i \rangle = \sum_i \langle u_i^2 \rangle = \frac{2}{3}k - 2\nu_T \frac{\partial \langle U_i \rangle}{\partial x_j} \tag{3.10}$$

In the realizable model, $\langle u_i^2 \rangle \left\langle u_j^2 \right\rangle - \langle u_i u_j \rangle^2 \geq 0$, which means this model is capable of performing better compared to the standard model, for the flows which have rotation and separation elements [31].

$k-\omega$ SST model : Here, k is the turbulent kinetic energy while ω stands for the specific dissipation rate and the term SST represents the shear stress transport model. The $k-\omega$ SST model, tries to combine the standard $k-\omega$ model and the $k-\epsilon$ model for the near wall region and the far field region respectively [32]. The equations 3.11 and 3.12 are used to model the k and ω respectively.

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial\left(\rho u_j k\right)}{\partial x_j} = P - \beta^* \rho \omega k + \frac{\partial}{\partial x_j}\left[\left(\mu + \sigma_k \mu_t\right)\frac{\partial k}{\partial x_j}\right] \tag{3.11}$$

$$\frac{\partial(\rho \omega)}{\partial t} + \frac{\partial\left(\rho u_j \omega\right)}{\partial x_j} = \frac{\gamma}{\nu_t} P - \beta \rho \omega^2 + \frac{\partial}{\partial x_j}\left[\left(\mu + \sigma_\omega \mu_t\right)\frac{\partial \omega}{\partial x_j}\right] + 2\left(1 - F_1\right)\frac{\rho \sigma_{\omega 2}}{\omega}\frac{\partial k}{\partial x_j}\frac{\partial \omega}{\partial x_j} \tag{3.12}$$

3.2.2. Near wall treatment

A precise estimation of the variables near wall region is important for the reliability of the solution as the gradients of these variable (pressure and velocity) experience sudden change. This estimation can lead to a better turbulence model [26]. In order to avoid the lager computational powers due to a very fine mesh near the wall regions, an approach called the wall functions is used.

3.2.2.1. Wall function

The wall functions are the formulas which create a connection between the variables like temperature, pressure and the velocity near the wall and at the wall. The turbulence variables are then formulated depending on the respective above mentioned turbulence models. Some of the commonly used wall functions are:

- Standard wall functions
- Non equilibrium wall functions
- Enhanced wall treatment

In this thesis, the enchanced wall treatment approach is used as this approach uses the concept of two layer zones which improves the modeling of wall bounded flows. This technique improves the prediction of wall shear stress and wall heat transfer by allowing the equations to be solved upto the wall [30].

Two layer zonal modeling In this modelling method, the domain is separated into two zones. These zones are identified by the wall distance which depends on the Reynolds number .

$$Re_y = y\frac{\sqrt{k}}{v} \tag{3.13}$$

In the equation 3.13, y represents the distance from the wall. The region affected by turbulence is defined for $Re_y > 200$ and for region affected by viscosity is $Re_y < 200$ [26]. For the region which is affected by viscosity, single equation turbulence model is applied for the kinetic energy and an algebraic term is used for the calculation of the energy dissipation rate. On the other head, for the turbulent region, a two-equation model is applied.

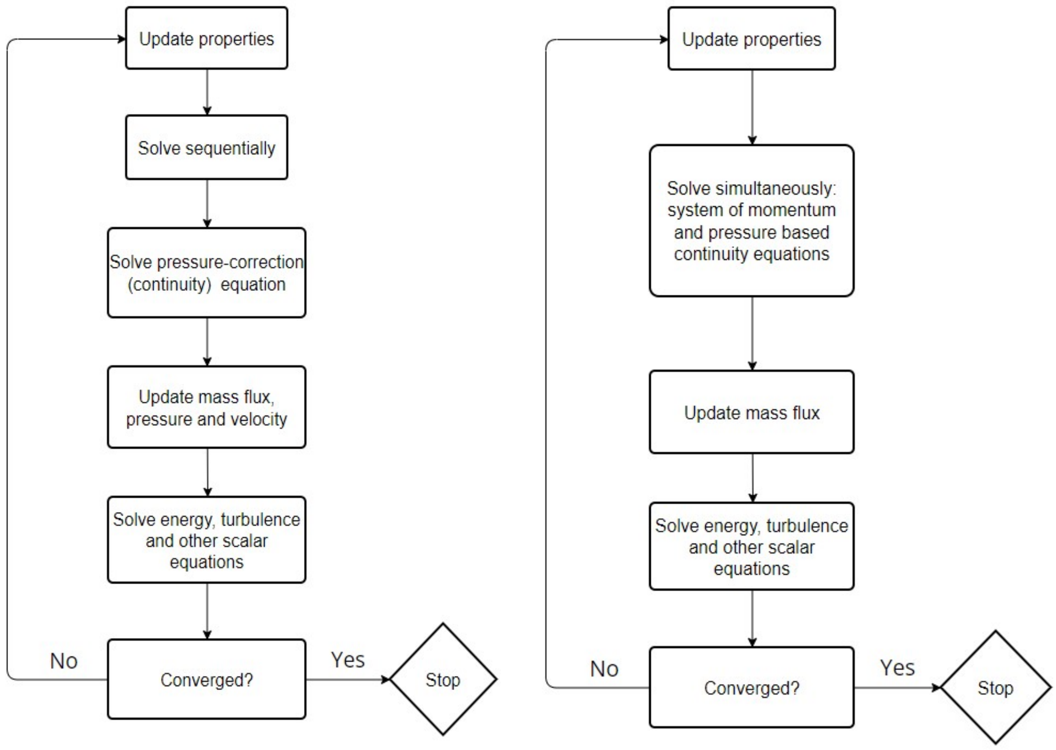
3.3. Solver type

Ansys Fluent uses two methods for the simulations, one of them is pressure based and the another is density based.

3.3.1. Pressure based solver type

The pressure based solver uses the projection method algorithm which means that this method solves the pressure correction equation to obtain the mass conservation limits for the velocity field [30]. As shown in figure 3.1, the pressure based solver provides two algorithms: A segregated algorithm and a coupled algorithm.

In the segregated algorithm, the governing equations are solved in a sequence (segregated). As the governing equations are non-linear and coupled, the iterative approach is used to obtain convergence. While for the coupled algorithm, it solves a coupled system of equations that consists of momentum and continuity equations [30].



(a) Pressure based segregated algorithm (b) Pressure based coupled algorithm

Figure 3.1.: Pressure-based solution algorithms

3.3.2. Density based solver type

The density based solver uses the concept of coupled algorithm which solves the governing equations simultaneously. Additionally, as shown in the figure 3.2, it uses the iterations to solve the equations until the convergence is achieved [30].

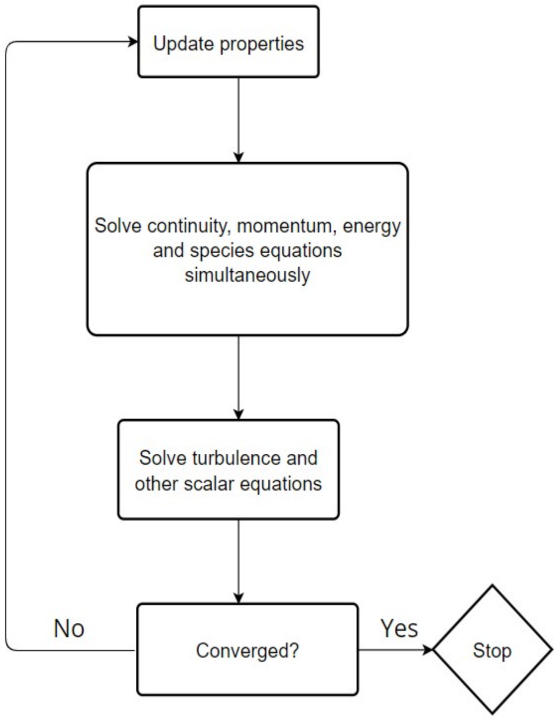


Figure 3.2.: Density based solution algorithm

CFD has proven to be a powerful tool for predicting the fluid flow behaviour in numerous applications. By applying the principles of fluid mechanics, numerical analysis, and computer science, CFD simulations can accurately model complex flow phenomena and provide insights into the behavior of fluid systems that are difficult or impossible to observe experimentally. The knowledge and skills acquired from this chapter will be essential for the subsequent chapter, which will apply CFD techniques to analyze and optimize specific fluid flow problems.

4. Research objectives

Hydrogen gas has set expectations of serving as a clean carrier of energy and has a lot of potential to be used as one of the primary energy carriers [1]. Although, there is a risk of serious explosion due to the properties of hydrogen (low ignition energy and high diffusivity). Once the process occurs, the exhaust gases that are being emitted have high temperatures. If there is an occurence of an energy source (even a small electrostatic spark discharge), the hydrogen gas (in combination with oxygen) has a tendency to readily ignite with that energy source and ultimately it might cause a fire or an explosion. Additionally, this unfortunate event of fire can spread rapidly and a risk of an accidental explosion is present. The gaseous explosions have the capability of causing serious damage to property and livelihood. Therefore, there is a requirement of designing a cooling system which aids in lowering the temperature of the flue gas upto the safety limits.

Studies have shown that in a few seconds of release, a large cloud of flammable gases could be formed. This cloud has the tendency of explosion due to the flammable and detonative nature of the hydrogen gas [33]. At high temperatures, hydrogen gas can rapidly expand, leading to increased pressure in containers or pipelines. If a leak occurs, the escaping hydrogen gas can form a flammable mixture with the surrounding air, and any spark or heat source can trigger an explosion. Hydrogen gas is more hazardous to deflagration-to-detonation transition (DDT) and is the most unwanted scenario which can occur due to an unexpected pre release event and the favourable conditions for the accident to take place can be termed as ATEX (Explosive Atomsphere), and this has to be avoided [34].

A possible solution to this problem is passing the hydrogen containing flue gas through a heat exchanger before releasing it to the atmosphere. The idea of using a heat exchanger can be a solution for lowering the temperature not only in this case but it can serve as a general solution. Flue gases containing different composition of gases can be cooled down using this approach. Additionally, tap water is available in abundance so it turns out to be a cost effective cooling agent.

A shell and tube heat exchanger is selected where a number of pipes are housed inside a shell and provides the surface for heat transfer. Arrangement of flow is in such a way that the flue gas flows inside the shell and the water is passed through the tubes in a counter flow scheme. The idea behind selecting shell and tube type heat exchanger is that it provides certain benefits over other kinds, such as: simple design, the construction is robust and relatively low maintenance costs. It's customizable design and efficient heat transfer capabilities make it a valuable equipment for industries including chemical processing, power generation and many others.

The aim of this thesis work is to develop, design (and optimize) a heat exchanger for explosive gases to cool them below self-iginition temperature.

From the above mentioned objectives, some of the research questions can be extracted. The following are the research question for the presented thesis work:

- **What is one possible solution for lowering the temperature of flue gas containing hydrogen?**
- **What factors should be considered while designing the heat exchanger? What is the procedure for sizing of the heat exchanger? What is the effect of the decisions made while sizing the heat exchanger with an economical point of view?**
- **How can the efficiency or effectivity of the heat exchanger can be improved without making a lot of changes in the design of the heat exchanger?**
- **A comparative study of the performance of classical shell and tube heat exchanger design and the shell and tube heat exchanger with modified design.**

The above mentioned research questions are answered in the following chapters. The first two questions are answered in the chapter 5 using the knowledge obtained from the state of the art while the remaining questions are answered in the chapter 6.

4.1. Methodology

The methodology of the thesis is presented here. The flow chart diagram 4.1, provides a brief idea about the work flow of the study.

While studying about various types of heat exchanger it is evident that the shell and tube heat exchanger is one of the best options available to cool down the exhaust gases due to it's sturdy and stable construction with less maintenance requirements. Additionally, there is a possibility of using different fluids in different cycles and the design of the heat exchanger is quite flexible. Hence, the thesis focuses on designing and simulating the shell and tube heat exchanger for the particular application.

The simulation was conducted using ANSYS Fluent, while the geometry of the shell and tube heat exchanger was first created using Autodesk Inventor, which allowed for the creation of the tubes, shell, and baffle plates. Two different models were created for the heat exchanger: one with normal baffles and another one with inclined baffles.

The idea behind creating two different models for the heat exchangers is to improve the effectivity and study the nature of the heat exchanger with an inclination of baffles. To ensure a proper comparison between the two models, the mesh parameters, boundary conditions and the material of the heat exchangers is kept same. With the tilt in the baffles, the nature of fluid flow changes in the vicinity of the baffles. This change of nature of fluid flow leads to different cooling effects and thus the effectivity of the heat exchanger might change.

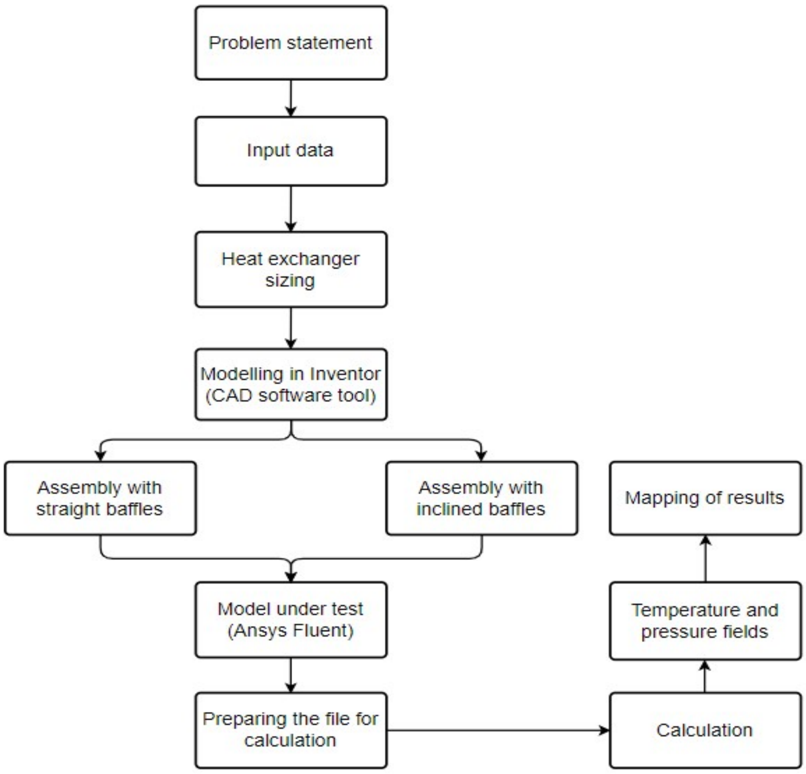


Figure 4.1.: Methodology

After the geometries were created, the models were meshed using the ANSYS Meshing tool. The meshing style was kept consistent between the two models to ensure a fair comparison. Boundary conditions were applied to the models based on the operating conditions of the heat exchanger. For the inlet, a uniform mass flow rate was applied. The fluid properties were defined as varying entities with the change in temperature, and the temperature for the inlet was obtained from the experimental data.

The models were then solved using ANSYS Fluent's solver, which uses the finite volume method to discretize the governing equations for fluid flow and heat transfer. The models were run until a steady-state solution was reached for both normal baffles and inclined baffles.

Finally, the results were analyzed and compared. The temperature distribution, velocity distribution, pressure drop, and heat transfer coefficient were calculated and compared between the two models. The analysis was conducted to determine if the inclined baffles provided any advantages over the normal baffles.

5. Design Considerations

The shell and tube heat exchanger has been designed using the Kern's method. While sizing the heat exchanger, a lot of factors and variables are needed to be considered. Usually, the sizing of the heat exchanger begins from either side (tube or shell) of the heat exchanger. The factors that must be considered to ensure it's optimal performance are listed below:

- **Heat transfer requirements:** It is essential to determine the amount of heat transfer that needs to occur from one fluid to another to achieve the desired temperatures. Calculating the heat transfer requirements involve the calculation of heat load and the logarithmic mean temperature difference.

- **Fluid properties:** The physical properties of the working fluids affect the rate of heat transfer and ultimately the sizing of the heat exchanger. These physical properties include specific heat capacity, the density of the fluids, viscosity and thermal conductivity.

- **Flow rates:** The heat transfer area required and the pressure drop are dependent on the flow rates of the fluids. Additionally, the fluid velocity of each side in the heat exchanger is important to ensure the heat transfer. If the velocities are too low, the fluid will not pass through the full course of the heat exchanger and back flow or negative pressure is observed.

- **Cost considerations:** Evaluation of the cost is important while designing any equipment as the cost includes, manufacturing cost, maintenance cost, installation cost etc. There are several ways where the cost can be effectively reduced without limiting the performance of the heat exchanger. One of the ways of designing an economical heat exchanger is to use longer tubes with less diameter, which would ultimately reduce the shell diameter as well. Longer tubes and a smaller diameter shell in a heat exchanger provide cost savings. For starters, it lowers building costs by utilizing less material. Second, it reduces weight and space needs, which saves money on shipping and installation. Third, it enhances heat transfer efficiency, possibly saving energy and lowering the size of connected equipment. Fourth, it lowers pressure drop, which saves energy in fluid circulation and lowers maintenance expenses. Finally, it offers scalability, flexibility, and ease of maintenance, resulting in less downtime and personnel expenditures. Overall, these considerations contribute to the low cost of heat exchangers with longer tubes and lower shell diameters.

To improve the efficiency of the heat exchanger, an effort has been made to modify the design of the heat exchanger by a small margin which involves tilting the baffles as the classical baffles have some demerits such as low heat transfer, dead zones where the stagnation of the fluid is observed. This study examines the influence of the tilted baffles on temperature, velocity and pressure distribution over the course of heat exchanger when the baffles are tilted at 30°.

5.1. CAD geometry

The CAD modelling of the shell and tube heat exchanger has been done in Autodesk Inventor. The software tool is used to create a 3D geometry which is further imported as a standard STEP (Standard for the Exchange of Product Data) file in Ansys Fluent for the CFD simulation purpose. The table 5.1, shows the dimension and quantity of the parts of the shell and tube heat exchanger assembly.

Table 5.1.: Design parameters

Parameter	Attribute
Shell diameter [mm]	80
Shell length [mm	700
Tube inner diameter [mm]	12
Tube outer diameter [mm]	16
Tube length [mm]	500
No. of Tubes	7
No. of Baffles	6
Angle of baffle inclination	30°

5.1.1. Heat exchanger assembly

In the figure 5.1, the CAD assembly of the heat exchanger is demonstrated. This assembly consists of 7 tubes, 6 baffles and 2 tube sheets enclosed by the shell.

To observe a better view of the inner part of the shell, a section view is shown in the figures 5.2 and 5.3. The figures 5.2 and 5.3 respresent the sectional views of standard baffles and inclined baffles respectively. From the sectional views, it is evident that the available pocket size between two consecutive baffles is different and the change in orientation of the baffles will lead to the change in flow pattern of the shell side fluid. There is no direct blocking of the fluid by the baffle which tends to reduce the dead zones inside the shell.

Figure 5.1.: Shell and tube heat exchanger CAD assembly

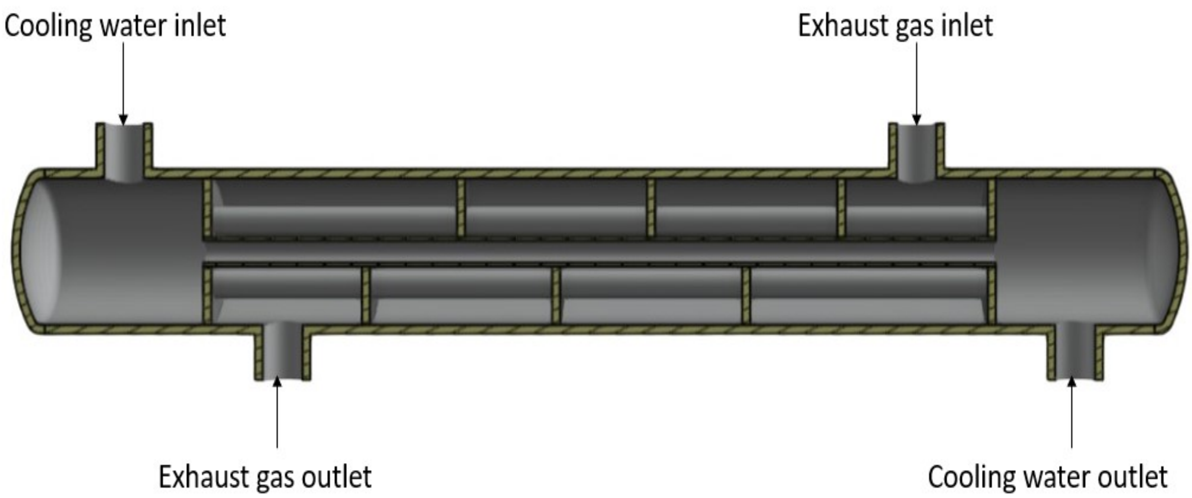


Figure 5.2.: Shell and tube heat exchanger cross section

Once the CAD assembly of the heat exchangers has been made, it is ready for further analysis of performance. This analysis is done to ensure the desired performance of the heat exchanger and is carried out in Ansys Fluent. As, the CAD part is done in Autodesk Inventor, the CAD file needs to be converted into a supportable file for CFD. The standard format used in Ansys is the STEP format which also enables Ansys to prepare the geometry for meshing. The steps involved in order to carry out a standard CFD procedure in Ansys Fluent are listed below.

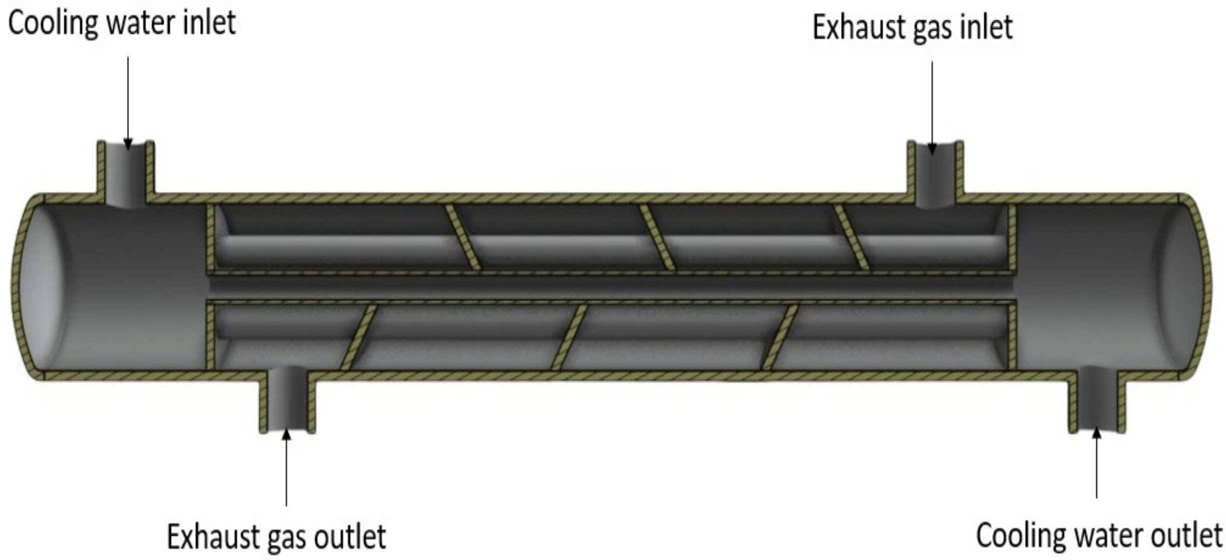


Figure 5.3.: Shell and tube heat exchanger cross section with inclined baffles

The first step in the Ansys environment is to make the CAD model ready for CFD simulation. It involves defining of the individual fluid domains so that the tool knows the solid and fluid domains. Once the fluids domains are defined, the next step is to define the inlet and outlet faces for the fluids. After successfully defining the geometrical data, the next step is to do the meshing of the model. Meshing is important as it divides the single model into number of small divisions which would be calculated. This is a critical step as the accuracy of the results depend highly on the mesh.

5.2. Mesh generation

ANSYS offers a number of meshing techniques that are appropriate for mesh creation in shell and tube heat exchangers. The meshing method used is determined by parameters such as geometrical complexity, required level of precision, computational efficiency, and specific characteristics to be captured inside the heat exchanger.

5.2.1. Sweep Meshing

Sweep meshing is a structured meshing approach offered in ANSYS that is especially well-suited for the tube-side domain of shell and tube heat exchangers. This approach extrudes 2D components down the tube axis, giving you control over mesh density, smoothness, and tube wall alignment. Sweep meshing captures boundary layer effects and properly mimics the flow behavior on the tube side [26].

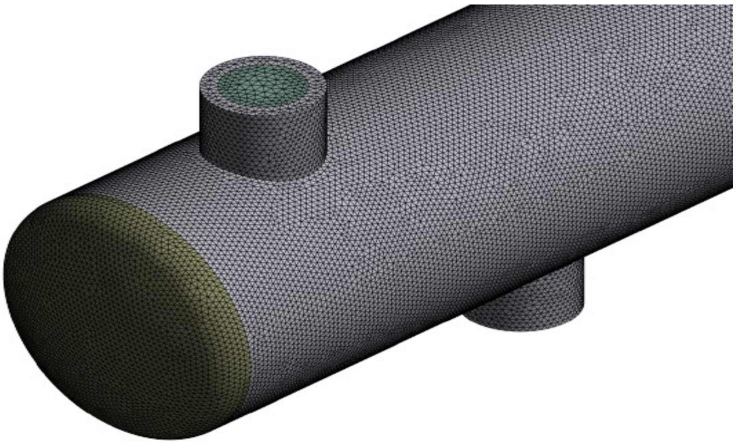


Figure 5.4.: Meshing of the Assembly

(a) Shell domain mesh for straight baffles

(b) Shell domain mesh for inclined baffles

Figure 5.5.: Meshing of the shell domain

5.2.2. Patch conforming Meshing

Patch conforming meshing, also known as mapped meshing, is another structured meshing approach used in shell and tube heat exchangers. To generate a conformal transition between the tube-side and shell-side domains, this approach employs 2D mapped face meshes. Patch conforming meshing properly simulates flow mixing, pressure loss, and heat transfer by preserving mesh connection throughout the interface.

5.2.3. Tetrahedral Meshing

Tetrahedral meshing might be useful for complicated geometries or irregular tube configurations. ANSYS provides a number of automated tetrahedral meshing techniques for producing high-quality meshes for the shell-side domain. The precise intricacies of the heat exchanger's geometry are captured by these tetrahedral pieces, allowing for realistic simulation of fluid flow behavior.

5.2.4. Inflation Layers

Meshing solutions for shell and tube heat exchangers rely heavily on inflation layers. These layers are used to correctly describe the flow behavior and heat transfer characteristics by capturing the boundary layer effects near solid surfaces such as tube walls. This section delves into the notion of inflating layers in mesh production, their significance, and their use in ANSYS software for shell and tube heat exchangers.

In computational fluid dynamics (CFD) simulations, inflation layers, also known as boundary layers or prism layers, are thin layers of mesh elements inserted close to solid surfaces. These layers are intended to resolve velocity and heat gradients inside the boundary layer, allowing for precise depiction of flow events near solid barriers [30].

The inflation layers create a fine mesh or the layers in the domain where the solid and fluid layers meet. It is useful for the boundary layer resolution in CFD as the high quality geometry aligned elements are capable of resolving the boundary layer growth. These layers can capture the normal gradient with minimum elements and provide accurate values of temperature and velocity gradients near the boundary walls of no-slip nature [26].

The table 5.2, shows the assigned mesh settings for the current mesh. The settings for both the models have been kept identical to have a better comparison of results. The results obtained by these mesh are in agreement with the calculated and desired results for both the cases. Once the meshing is completed, next stage in the CFD process is assigning the boundary conditions.

The boundary conditions includes different assignments. They are listed below:

- **Materials:** Assigning the materials of the shell and tube, each domains, the working fluids is done here in the third step.
- **Turbulence and energy model:** According to the nature of the flow, the turbulence model and the energy models are assigned in this section of Ansys, which is essential for the calculation of the results. Different turbulent models are selected according to the type of application.
- **Solver:** The type of solver is also assigned right before the calculation.
- **Convergence criterion:** This option informs the solver regarding different convergence conditions and the solver acts accordingly.

Table 5.2.: Mesh parameters

Parameter	Attribute
Physics preference	CFD
Element order	Linear
Element size	0.3 m
Growth rate	1.2
Mesh defeaturing	Yes
Defeature size	$1.5e-003$ m
Capture Curvature	Yes
Curvature Min Size	$3e-003$ m
Curvature Normal Angle	18°
Capture Proximity	Yes
Proximity Min Size	$3e-003$ m
Proximity Gap Factor	3
Inflation	Yes
Inflation Option	Smooth Transition
Inflation Algorithm	Pre

- **Iterations:** No. of iterations are defined at last which informs the solver to solve the equations as many times until the convergence has been achieved.

After assigning these parameters, finally the solving starts in Ansys and once the convergence has been achieved, finally the results are plotted which can be seen in the following chapter.

6. Results

In this chapter the findings of the numerical calculation using the knowledge of CFD obtained from the chapter 3. The results are described in the form of contour plots. The plots describe the fluid properties (Temperature, velocity and pressure) for the shell side i.e. the exhaust gas. The gas enters from the right top inlet port and on the left side the outlet port is placed at the bottom.

6.1. Temperature contour

As observed in the figure 6.1 , a distribution of temperature through a contour explains the thermal behaviour of the heat exchanger. The observed termperature value of the exhaust gas when it exits from the heat exchanger, after the heat transfer has occoured, is 533K (260°C). While for water, which is the cooling agent, the measured temperature value at the tube outlet is 340K (67°C).

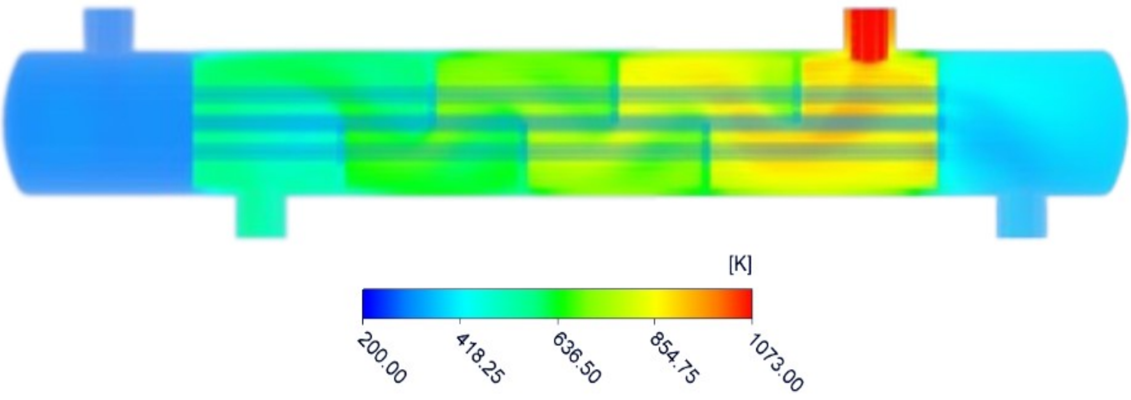


Figure 6.1.: Temperature contour of the heat exchanger

6.2. Velocity contour

As observed in the figure 6.2 , a distribution of velocity of the fluid particles is described. The observed average velocity of the exhaust gas when it exits from the heat exchanger, after the heat transfer has occoured, is 36.78 m/s. While for water, which is the cooling agent, the measured temperature value at the tube outlet is 2.94 m/s.

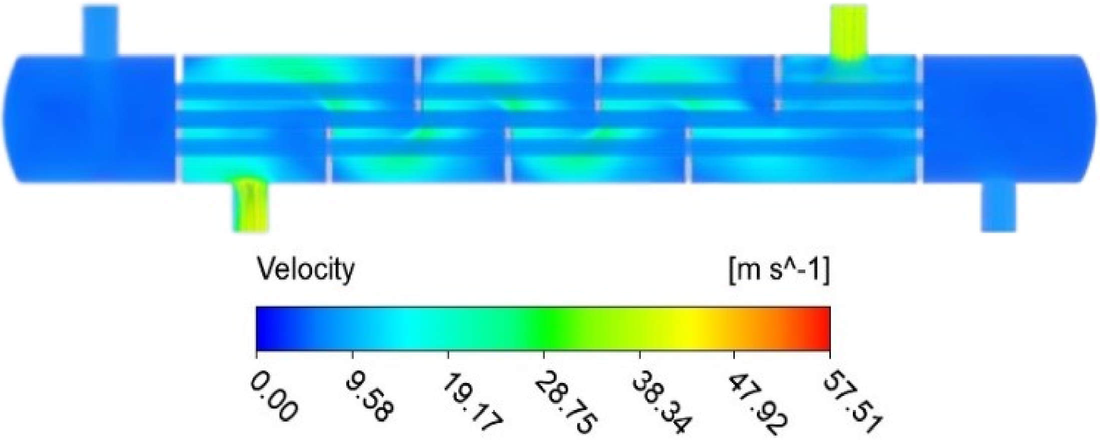


Figure 6.2.: Velocity contour of the heat exchanger

An effort to improve the effectivity of the heat exchanger has been made by changing the orientation of the baffles. In the classic design, the baffles are perpendicularly aligned with the tubes and the shell. While in the second design, a tilt angle of 30° is provided and the same study is carried out to compare the results.

A comparison of the heat exchanger models with straight and inclined models have been made in the results to examine the effect of inclination of the baffles on the fluid characteristics. With the inclination of baffles, the available area for the heat transfer changes and also a change in the flow pattern is observed. Due to this phenomenon, the mean velocity of the flow and turbulence is affected which further affects the heat transfer. In order to examine the change, a comparison for the properties such as temperature, veloctiy and pressure has been made for both the models which are seen below.

6.3. Temperature plots comparison

The temperature distribution along the heat exchanger can be observed through the side view on the symmetrical plane. The figure 6.3 shows the comparison of the temperature contour of the heat exchanger model with straight and inclined baffles and the figure 6.4 provides the information about the shell wall temperature.

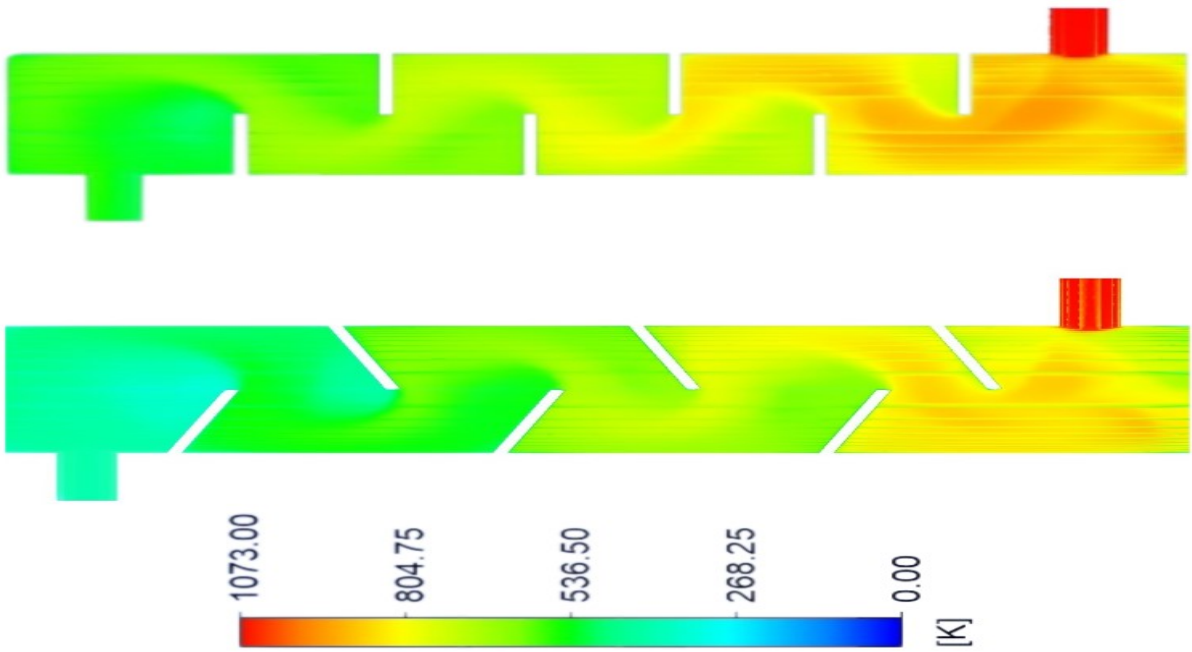


Figure 6.3.: Temperature contour comparison of both models

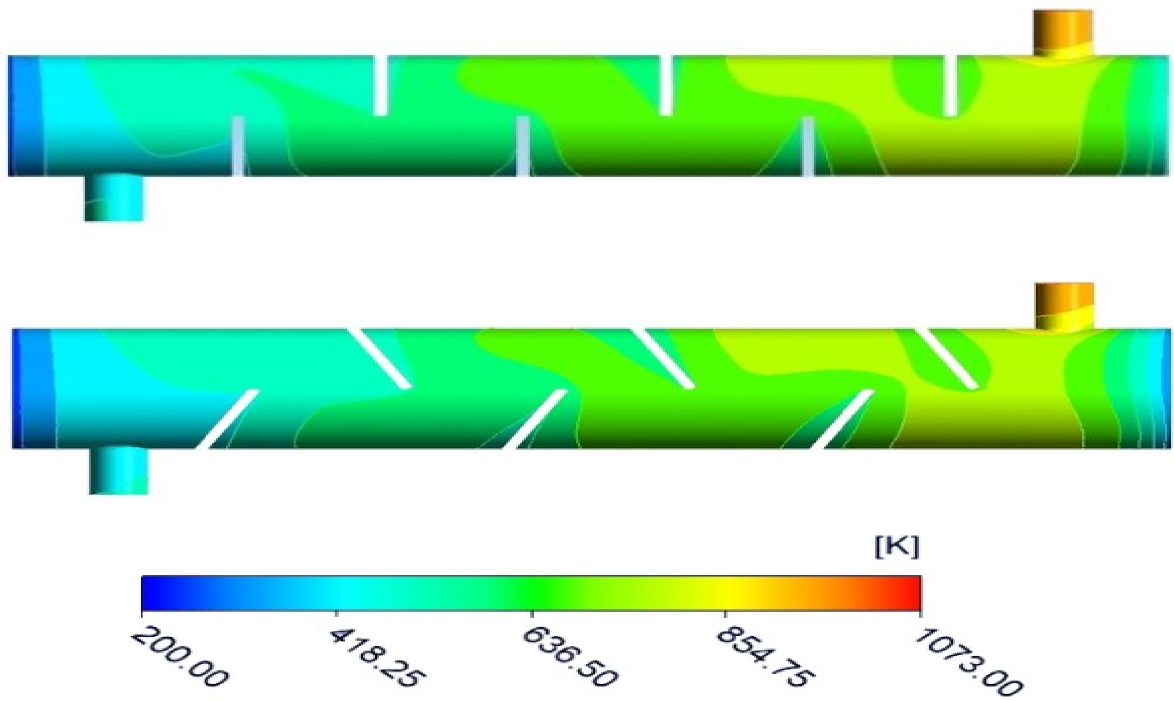


Figure 6.4.: Shell wall temperature comparison of both models

While comparing the two models, it is observed that the orientation of the baffles can have a significant impact on the fluid flow and hence the heat transfer. As observed from the figures 6.3 and 6.4, the cooling effect is better in the heat exchanger that has the inclined baffles. In the figure 6.3, it is observed that the mean flow of the hot fluid near the open ends of the baffle is getting cooler as it is passing by each baffle. This particular frame of observation indicates better cooling performance in the case of inclined baffles. The reason behind this behaviour is the inclination of the baffles cause more turbulence in the flow when compared to the straight baffles which further enables better mixing of the fluid resulting in increase of the contact between the fluid and the heat transfer surface. Additionally, the inclination of the baffles create a uniform flow distribution that reduces the possibility of hot spots and cold spots.

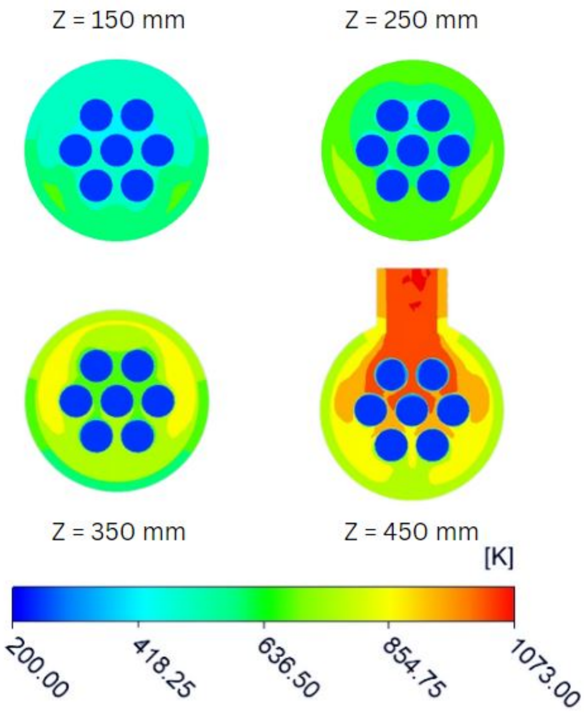


Figure 6.5.: Sectional view at different locations

The figures 6.5 and 6.6 describes the temperature contours for different cross sections along the length of the heat exchanger. It helps in examining the nature of flow inside the shell domain at different locations and the effectiveness of the calculated shell diameter. Additionally, the information of temperature at different cross sections, helps in identifying the areas of the heat exchanger that experience temperature gradients which further shows the areas of poor heat transfer or the areas with potential hotspots.

The figure 6.5 represents the information for the heat exchanger model with straight baffles while the figure 6.6 is for the heat exchanger model with the inclined baffles. In both the models, the cross sections are taken at the distance of Z = 150, 250, 350 and 450 mm along

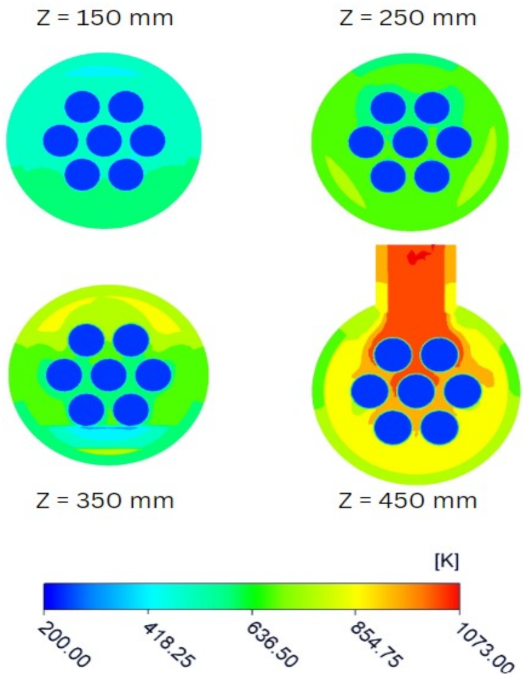


Figure 6.6.: Sectional view at different locations

the length of the heat exchanger where Z = 0 represents the inlet location of the cold fluid.

Comparing both the images, it is observed that the temperature distribution in the case of the inclined baffles is uniform while in the case of heat exchanger with straight baffles, the ares of hotspots are observed. Moreover, the cooling effect or the heat exchange appears to be much efficient in the heat exchanger that has inclined baffles.

6.4. Velocity plots comparison

The figure 6.7 describes the mean velocity of the shell domain. When comparing the two models, it is evident that the orientation of the baffles play a significant role on the mean velocity. In case of the straight baffles, the flow is comparatively less turbulent which results into lesser velocity values of the fluid.

The flow when observed right after when it passes through the baffles, indicates that the inclination of the baffles aids in attaining higher velocities and the distribution of the flow is also uniform. Comparing the mean axes of flow and baffles, in case of inclined baffles, it is parallel. This helps in reducing the stagnant zones right behind the baffles and which in turn improves the heat transfer area resulting into better effectiveness of the heat exchanger.

From the figure 6.8, it is evident that the nature of the flow in the second case is able to

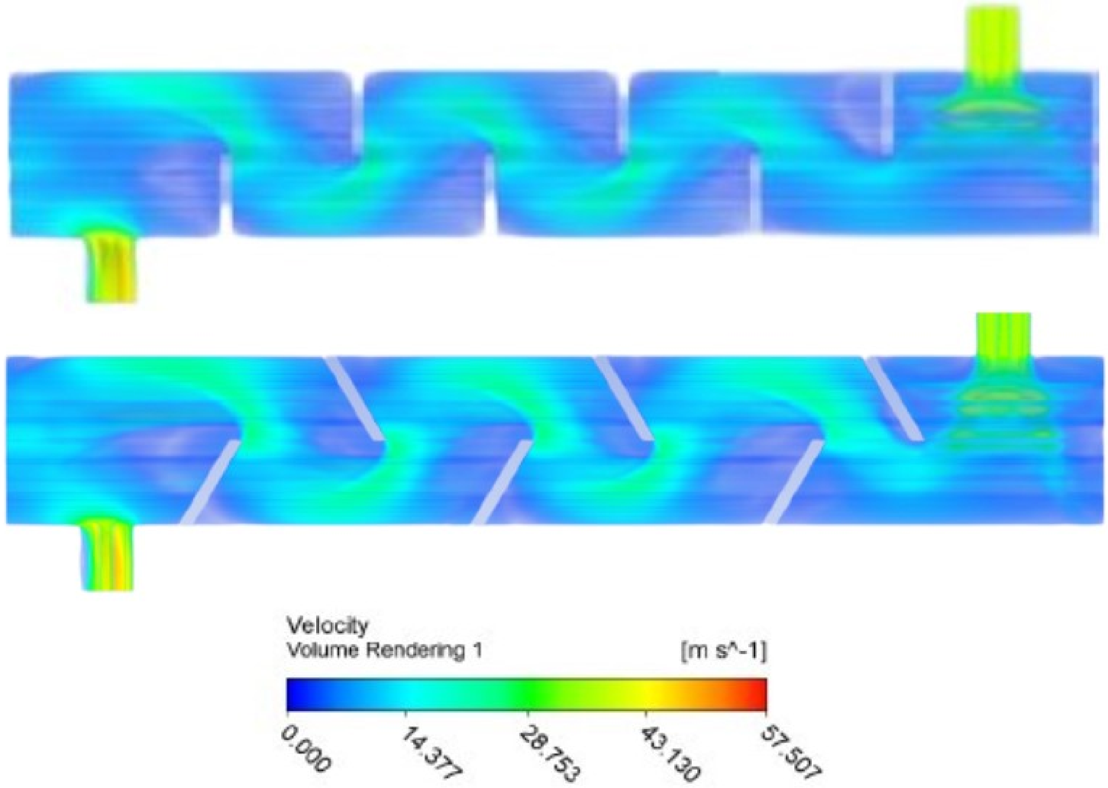


Figure 6.7.: Velocity comparison of both models

attain high velocities just after passing through the baffle and the vector. While in the case of straight baffles, a large velocity gradient is not present which states that it has less turbulence compared to the inclined baffles. The impact of mean velocity is of great importance when it comes to heat transfer as the mean velocity is in direct proportion of Reynolds number (ultimately turbulence).

To understand the flow direction and pattern, figure 6.9 shows a comparison of the formed streamline by each fluid particle tracing the flow. As observed in the heat exchanger model with straight baffles, there are formation of the stagnation zones which hinder the process of heat transfer by reducing the Reynolds number and these zones are right behind the baffles.

In the later model, the arrangement of the inclined baffles has been done in such a way that it aids the flow. The baffles have been placed in opposite direction in the upper and lower halves. This type of arrangement guides the flow with higher velocities and ensuring that there are no or minimum stagnation zones. Additionally, there is lesser volume available between two consecutive baffles which also helps in increasing the velocity of the fluid flow.

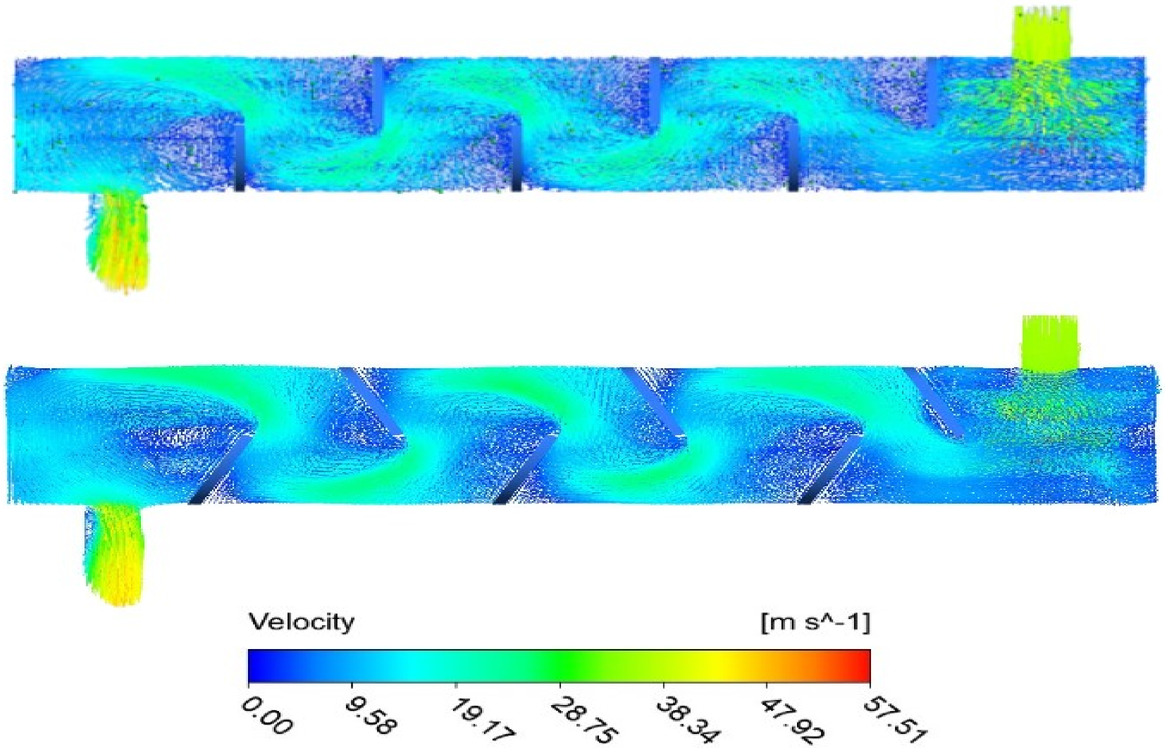


Figure 6.8.: Velocity vector comparison of both models

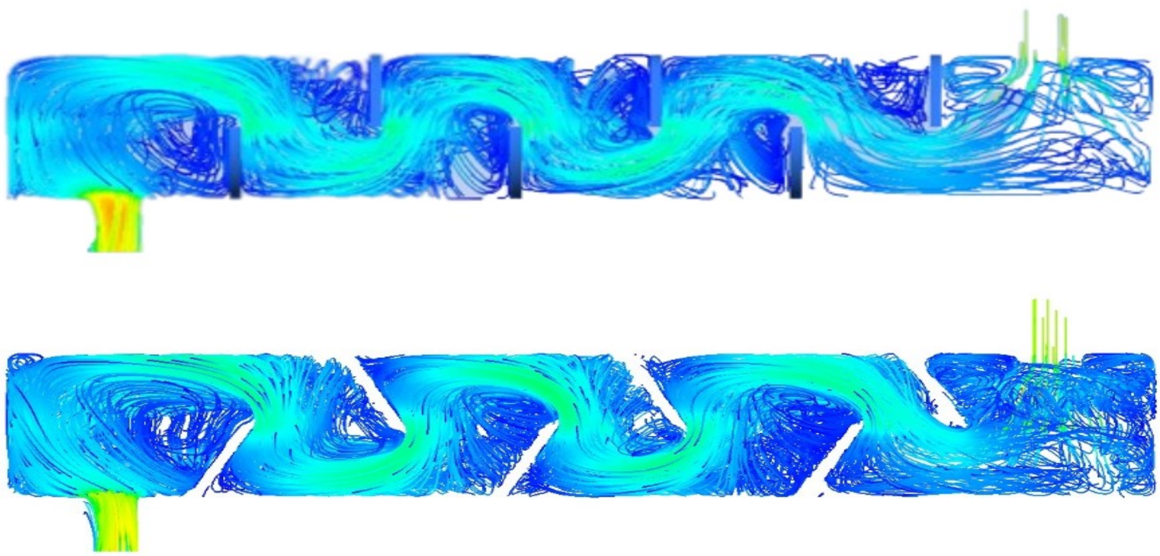


Figure 6.9.: Streamline comparison of both models

6.5. Pressure plots comparison

Pressure drop can be defined as the difference of pressure between the inlet and outlet ports of the heat exchanger. It is an important criterion in the design and operation of the heat exchanger as it might affect the overall efficiency of the heat exchanger.

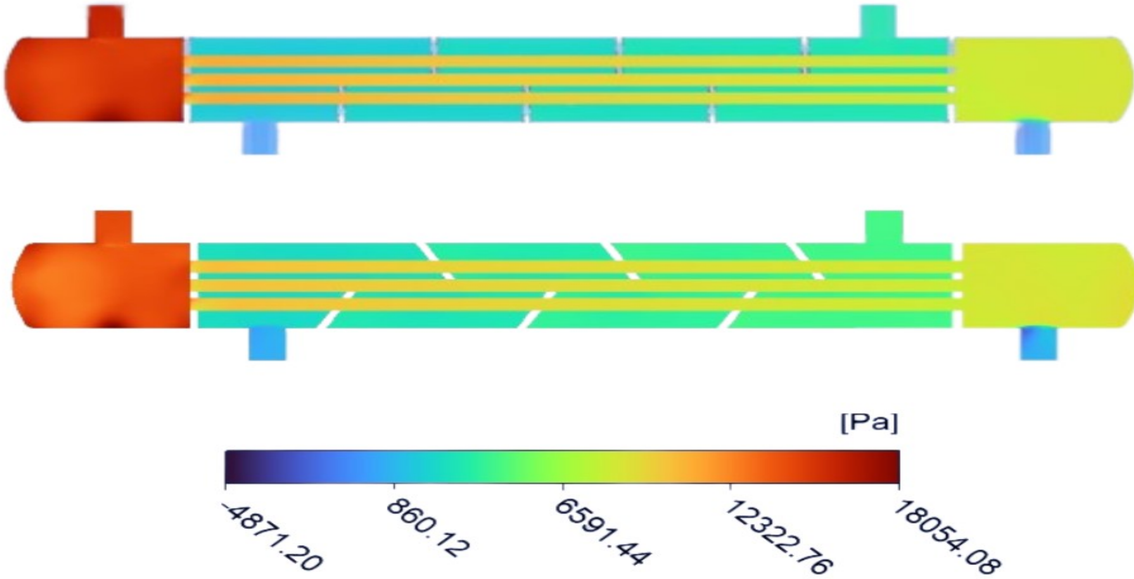


Figure 6.10.: Pressure comparison of both models

While comparing the heat exchanger models, it is very important to understand the impact of baffle orientation on pressure drop. Normally, when the baffles are inclined at a certain angle, a higher pressure drop is observed compared to straight baffles because of the mixing of the fluid flow and increased turbulence. However, this increased pressure drop can lead to more efficient heat transfer because the increased turbulence and the mixing of fluid results in thinner boundary layer. In the close vicinity of the baffles, major pressure gradient is observed in the case of inclined arrrangement due to higher resistance experienced by the fluid because of turbulent mixing.

The effect of the baffles on pressure drop depends on various factors but generally, a slight increase in pressure drop can be observed to to inclined arrangement of the baffles but this increased pressure drop can have both effects (negative and positive) on the efficiency and effectiveness of the heat transfer process.

7. Conclusion and Future Work

7.1. Conclusion

To conclude, this chapter describes a comprehensive analysis of the hazards associated with the hydrogen gas. The potential promised in hydrogen as an energy carrier is indisputable, but the features, such as low ignition energy and strong diffusivity, necessitate careful consideration for safer use. In the context of flue gas that contains hydrogen, an investigation has been made in the requirements for a cooling system to avoid the explosion hazards arising due to the above mentioned characterisitcs of the hydrogen gas. The flammable and detonative nature of hydrogen has a tendency to undergo deflagration-to-detonation transition, which is hazardous. The occurence of such explosive atmospheres, also known as ATEX, must be strictly avoided. The notion of heat exchanger has been addressed in the above chapters which turn out to be one of the most effective solutions to cater the problem.

In this thesis work, a cooling system has been developed in the form of a shell and tube heat exchanger to cool down the exhaust gases from a temperature of 800°C. The coolant that has been used is the running tap water as it is available in abundance and possess good cooling properties. The shell and tube heat exchanger has some advantages over other types of the heat exchangers, such as sturdy construction, flexible design and effective heat transfer capabilities. Considering the economical point of view, the aspect ratio of the heat exchanger is kept in such a way that the length of the heat exchanger in given priority and the diameter is kept as low as possible. This ensures low cost and high velocities which improve the performance of the heat exchangers.

The sizing of the heat exchanger has been done using the Kern method which proves to be a good iterative method and it has some standard corelations for the initial stage of sizing. The modelling of the heat exchanger assemly is carried out using Autodesk Inventor 2021. Initially, a conventional model of the heat exchanger is studied and then, the design is modified to inclined baffles to have a comparison for the properties such as heat transfer, pressure and velocity. For the comparison, each of the parameters such as the mesh type and number of elements, inlet mass flow rate and temperature, type of turbulence model, the solver and other boundary conditions were kept same.

Ansys Fluent v2022 R1 has been used to carry out the CFD simulations using the stand alone components. For preparing the assembly to do the calculation, Spaceclaim is used from the Ansys workbench. The calculations were performed using the realisable $k - \epsilon$ turbulence model which assumes that the turbulent flow can be modelled using two differnet equations for the turbulent kinetic energy and the dissipation rate. For this particular project, $k-\epsilon$ is more suitable to calculate the flow and heat transfer phenomenon rather than the $k-\omega$ model as the flow did not have a significant rotational effect.

The velocity plots, as shown in figure 6.7, illustrates that the mean velocity flow on the shell side for the conventional model has a slight gradient when it passes through the baffle while in the case of the modified design, the same flow experiences better gradient and the mean flow velocity seems to be increased right after when it passes the open end of the baffle. The average value of the mean flow velocity in the first model is in the range of 10-16 m/s while in the second model, it is evident that the velocity range extends upto 24 m/s for a certain period. This is due to the better guidance for the flow and change in the angle of attack for the obstruction as shown in figures 6.8 and 6.9. Additionally, the increased uniformity of the flow results into thinner boundary layers in the vicinty of the baffles, this helps in achieving higher velocities.

If the pressure plots are compared for both the models, it is observed that the heat exchanger model with inclined baffles experiences higher pressure drop in an attempt to achieve higher turbulence by better mixing of the fluid.

The CFD results showed that the heat exchanger with the conventional design is capable of cooling the exhaust gas upto 268°C while the heat exchanger with inclined baffles cooled down the same gas upto 249°C. Additionally, the temperature contours showed that the model with inclined baffles had a more uniform temperature distribution across the length and diameter of the heat exchanger. The plots indicate a more efficient heat transfer in the heat exchanger with inclined baffles and an enhanced cooling effect. The figures 6.5 and 6.6 show the sectional views at certain locations which illustrates the better cooling effect of the modified design. It is observed that the inclined baffles provide better mixing of the fluid which results into higher turbulence in the flow. This facilitates better heat transfer phenomenon and a greater cooling effect is achieved.

Finally, heat exchangers with conventional and inclined baffles have different benefits and drawbacks. The standard baffle configuration is simple and extensively used. Turbulence in fluid flow improves heat transmission and performance. It may have increased pressure drop and heat transfer limits. The inclined baffle design has promising heat transfer properties. Inclined baffles make the flow route more convoluted, improving mixing and heat transfer. In heat transfer-critical conditions, this design may improve performance. The tilted baffles may complicate fabrication and maintenance. Considering heat transfer efficiency, pressure drop constraints, fabrication convenience, and maintenance, conventional and inclined baffles should be chosen based on application requirements. Optimizing heat exchanger performance in industrial applications requires assessing these aspects.

7.2. Future Works

The shell and tube heat exchangers are an essential part of the process industry and there is still a lot of research going on to improve the efficiency of the heat exchangers. Here are some of the possible future scopes of research in the betterment of the shell and tube heat exchangers.

- Carrying out simulations for different inclination angle: In the presented work, the comparison is made with the inclination angle of 30° but there is a scope of comparison by inclining the baffles at different angle and studying the behavior of the fluid properties. This will help the in finding the most suitable configuration settings for best results and thereby, increasing the efficieny.

- Limiting pressure drop: Finding the right angle of inclination which limits the pressure drop would be useful in terms of designing the heat exchanfer economics and performance. With the change in tilt angle, the pressure drop and the heat transfer changes, finding a right balance in the variables is essential. For this, a series of models having different inclination must be formed and simulated to assess the behaviour. Here, the DOE can be utilized by designing the model in such a way that Ansys is able to form different models automatically.

- Optimize the fluid flow pattern: The fluid flow pattern can be optimized by providing some flow guiding mechanism which enables better tubulent conditions for the efficient heat transfer to take place. However, a lot of research is needed to develop new flow patterns, may be spiral or wavy, which deliver the desired task feasibly.

- Experimenting with the pipe geometry: The pipe geometry, that is the cross-section of the pipe can be changed from circular to a different shape to study the effects of that shape on the fluid flow pattern. Some of the shapes are oval cross-section, spirular or wavy pattern to change the flow area and the available heat transfer area. This can lead to changed turbulence on the tube side.

- Application of artificial intelligence: The aid of artifical intelligence can be utilized to optimize the heat exchanger in terms of flow patterns, heat transfer and the design of the heat exchanger. Research is still needed to make a conceptualized model of heat exchanger using AI which has the capability of enhancing the operation and design.

- The current thesis may be used to develop an experimental method for designing and producing a shell and tube heat exchanger. Following STHX construction, a series of experiments may be carried out to determine the final heat transfer coefficient and Reynolds number on the shell and tube sides. These data can then be compared using the spreadsheet tool for additional confirmation and research.

Bibliography

[1] Woo Kyung Kim, Toshio Mogi, and Ritsu Dobashi. "Fundamental study on accidental explosion behavior of hydrogen-air mixtures in an open space". In: *International Journal of Hydrogen Energy* 38.19 (2013), pp. 8024–8029. ISSN: 03603199. DOI: 10.1016/j.ijhydene.2013.03.101.

[2] R. K. Shah and Duésan P. Sekuliâc. *Fundamentals of heat exchanger design*. Hoboken, NJ: John Wiley & Sons, 2012. ISBN: 8126538503.

[3] *VDI Heat Atlas*. Berlin, Heidelberg: Springer Berlin Heidelberg, 2010. ISBN: 978-3-540-77876-9. DOI: 10.1007/978-3-540-77877-6.

[4] Francesca Ceglia et al. "Modelling of Polymeric Shell and Tube Heat Exchangers for Low-Medium Temperature Geothermal Applications". In: *Energies* 13.11 (2020), p. 2737. DOI: 10.3390/en13112737.

[5] Francois P.A. Prinsloo, Jaco Dirker, and Josua P. Meyer. "Heat Transfer and Pressure Drop Characteristics in the Annuli of Tube-in-Tube Heat Exchangers (Horizontal Lay-Out)". In: *Proceedings of the 15th International Heat Transfer Conference*. Connecticut: Begellhouse, 2014. ISBN: 978-1-56700-421-2. DOI: 10.1615/IHTC15.FCV.009225.

[6] S. Kakaç and A. Pramuanjaroenkij. *Heat exchangers: Selection, rating, and thermal design / Sadık Kakaç, Hongtan Liu, Anchasa Pramuanjaroenkij*. 3rd ed. Boca Raton, FL: CRC Press, 2012. ISBN: 1439849900.

[7] Muhammad Imran, Nugroho Agung Pambudi, and Muhammad Farooq. "Thermal and hydraulic optimization of plate heat exchanger using multi objective genetic algorithm". In: *Case Studies in Thermal Engineering* 10 (2017), pp. 570–578. ISSN: 2214157X. DOI: 10.1016/j.csite.2017.10.003.

[8] Michele Zehnder. *Efficient air-water heat pumps for high temperature lift residential heating, including oil migration aspects*. DOI: 10.5075/EPFL-THESIS-2998.

[9] Bohong Wang et al. "Heat exchanger network retrofit with heat exchanger and material type selection: A review and a novel method". In: *Renewable and Sustainable Energy Reviews* 138 (2021), p. 110479. ISSN: 13640321. DOI: 10.1016/j.rser.2020.110479.

[10] Ramesh K. Shah and Duan P. Sekuli, eds. *Fundamentals of Heat Exchanger Design*. Hoboken, NJ, USA: John Wiley & Sons, Inc, 2003. ISBN: 9780470172605. DOI: 10.1002/9780470172605.

[11] *U-Tube Heat Exchanger | Cixi Fly Pipe Equipment Co.,Ltd*. 2019. URL: https://www.nbfvp.com/u-tube-heat-exchanger/.

[12] Günther Kirchner. "O1 Hints on the Construction of Heat Exchangers". In: *VDI Heat Atlas*. Berlin, Heidelberg: Springer Berlin Heidelberg, 2010, pp. 1523–1552. ISBN: 978-3-540-77876-9. DOI: 10.1007/978-3-540-77877-6$\backslash$textunderscore.

[13] T. Kuppan. *Heat exchanger design handbook*. Vol. 126. Mechanical engineering. New York: Marcel Dekker, 2000. ISBN: 9780824797874.

[14] Kenneth J. Bell. "Introduction". In: *HEDH Multimedia*. Begellhouse, 2014. ISBN: 978-1-56700-423-6. DOI: 10.1615/hedhme.a.015655.

[15] Rajiv Mukherjee et al. "Effectively design shell-and-tube heat exchangers". In: *Chemical Engineering Progress* 94.2 (1998), pp. 21–37.

[16] Anand Jayant Kulkarni. *Modified Probability Collectives Approach for Solving Engineering Problems*. DOI: 10.13140/RG.2.2.33706.82881.

[17] A. K. Golder. "Process Design of Heat Exchanger: Types of Heat exchanger, process design of shell and tube heat exchanger, condenser, and reboilers". In: *Chemical Engineering* 41 ().

[18] Ann Marie Flynn, Toshihiro Akashige, and Louis Theodore. *Kern's Process Heat Transfer*. Second edition. Hoboken, NJ: Wiley, 2019. ISBN: 978-1-119-36482-5.

[19] Donald Q. Kern. *Process heat transfer*. McGraw-Hill classic textbook reissue series. New York: McGraw-Hill, 1990. ISBN: 0070341907.

[20] J. M. Coulson and J. F. Richardson. *Coulson & Richardson's chemical engineering*. Oxford and Boston: Butterworth-Heinemann, 1996-. ISBN: 0750644443. URL: http://www.loc.gov/catdir/description/els032/95023187.html.

[21] P. Jungbecker and D. Veit. "Computational fluid dynamics (CFD) and its application to textile technology". In: *Simulation in Textile Technology*. Elsevier, 2012, 142–178e. ISBN: 9780857090294. DOI: 10.1533/9780857097088.142.

[22] Bengt Sundén. "Computational Heat Transfer in Heat Exchangers". In: *Heat Transfer Engineering* 28.11 (2007), pp. 895–897. ISSN: 0145-7632. DOI: 10.1080/01457630701421661.

[23] M. Prithiviraj and M. J. Andrews. "THREE DIMENSIONAL NUMERICAL SIMULATION OF SHELL-AND-TUBE HEAT EXCHANGERS. PART I: FOUNDATION AND FLUID MECHANICS". In: *Numerical Heat Transfer, Part A: Applications* 33.8 (1998), pp. 799–816. ISSN: 1040-7782. DOI: 10.1080/10407789808913967.

[24] Muhammad Mahmood Aslam Bhutta et al. "CFD applications in various heat exchangers design: A review". In: *Applied Thermal Engineering* 32 (2012), pp. 1–12. ISSN: 13594311. DOI: 10.1016/j.applthermaleng.2011.09.001.

[25] M. V. Volkenshteĭn. *Entropy and information*. Vol. v. 57. Progress in mathematical physics. Basel: Birkhauser, 2009. ISBN: 303460078X.

[26] *ANSYS FLUENT 12.0 User's Guide*. URL: https://www.afs.enea.it/project/neptunius/docs/fluent/html/ug/main_pre.htm.

[27] Bengt Andersson. *Computational fluid dynamics for engineers*. Cambridge and New York: Cambridge University Press, 2012. ISBN: 978-1-107-01895-2.

[28] N. Ashton et al. "Assessment of RANS and DES methods for realistic automotive models". In: *Computers & Fluids* 128 (2016), pp. 1–15. ISSN: 00457930. DOI: 10.1016/j.compfluid.2016.01.008.

[29] Nor Azwadi Che Sidik et al. "A Short Review on RANS Turbulence Models". In: *CFD Letters* 12.11 (2020), pp. 83–96. DOI: 10.37934/cfdl.12.11.8396.

[30] *ANSYS Fluent Theory Guide*. URL: http://www.pmt.usp.br/academic/martoran/notasmodelosgrad/ANSYS%20Fluent%20Theory%20Guide%2015.pdf.

[31] H. K. Versteeg and W. Malalasekera. *An introduction to computational fluid dynamics: The finite volume method / H.K. Versteeg and W. Malalasekera*. 2nd ed. Harlow: Prentice Hall, 2007. ISBN: 978-0-13-127498-3.

[32] F. R. Menter. "Two-equation eddy-viscosity turbulence models for engineering applications". In: *AIAA Journal* 32.8 (1994), pp. 1598–1605. ISSN: 0001-1452. DOI: 10.2514/3.12149.

[33] Zine labidine Messaoudani et al. "Hazards, safety and knowledge gaps on hydrogen transmission via natural gas grid: A critical review". In: *International Journal of Hydrogen Energy* 41.39 (2016), pp. 17511–17525. ISSN: 03603199. DOI: 10.1016/j.ijhydene.2016.07.171.

[34] *LP 2013, 14th International Symposium on Loss Prevention and Safety Promotion in the Process Industries: 12 - 15 May 2013, Florence, Italy*. Vol. 31. Chemical engineering transactions. Milano: AIDIC, 2013. ISBN: 978-88-95608-22-8.