Thermal performance analysis of passive techniques in shell and tube heat exchanger (STHXs): A CFD Approach

DISSERTATION

Submitted in Partial Fulfillment of the

Requirement for Award of the Degree of

MASTER OF TECHNOLOGY

IN

MECHANICAL ENGINEERING

By

Ravi Kumar Sen

(11004233)

DEPARTMENT OF MECHANICAL ENGINEERING

LOVELY PROFESSIONAL UNIVERSITY

PHAGWARA, PUNJAB (INDIA) -144402

2015

Thermal performance analysis of passive techniques in shell and tube heat exchanger (STHXs): A CFD Approach

DISSERTATION

Submitted in Partial Fulfillment of the

Requirement for Award of the Degree of

MASTER OF TECHNOLOGY

In

MECHANICAL ENGINEERING

By

RAVI KUMAR SEN

(11004233)

Under the Guidance of

MR. SUDHANSHU DOGRA

DEPARTMENT OF MECHANICAL ENGINEERING

LOVELY PROFESSIONAL UNIVERSITY

PHAGWARA, PUNJAB (INDIA) -144402

2015

Lovely Professional University Jalandhar, Punjab

CERTIFICATE

I hereby certify that the work which is being presented in the Dissertation entitled **"Thermal performance analysis of passive techniques in shell and tube heat exchanger (STHXs): A CFD Approach"** in partial fulfillment of the requirement for the award of degree of **Master of Technology** and submitted in Department of Mechanical Engineering, Lovely Professional University, Punjab is an authentic record of my own work carried out during period of Dissertation under the supervision of **Mr. Sudhanshu Dogra**, **Assistant Professor**, Department of Mechanical Engineering, Lovely Professional University, Punjab.

The matter presented in this dissertation has not been submitted by me anywhere for the award of any other degree or to any other institute.

.

Date: **(Ravi Kumar Sen)**

This is to certify that the above statement made by the candidate is correct to best of my knowledge.

Date: **(Mr. Sudhanshu Dogra)**

Supervisor

The M-Tech Dissertation examination of Ravi Kumar Sen, has been held on date __________

Signature of Examiner

ACKNOWLEDGEMENT

I take this opportunity to express my deep sense of gratitude towards my guide Mr. Sudhanshu Dogra for his expert guidance during the preparation of this project.

I am also thankful to my university which assigned me this project which immensely improved our technical knowledge. I am thankful to university for providing me various resources like lab, tools for the timely completion of this project.

I also express to my thanks to all of them directly or indirectly have helped me in preparing this project. Last but not the least I would like to thank family and friends for their boost and support in every sphere. Their vital push infused a sense of insurgency in me.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

ABSTRACT

In the present study, 3D CFD simulations are carried out to investigate the effects of lumped tube on shell-side flow and heat transfer of a shell-and-tube heat exchanger (STHX) with helical baffles. Passive techniques of shell and tube heat exchanger have been investigated by a CFD approach, which is a useful tool in fluid flow analysis. Shell and tube heat exchanger is widely used in industrial application like oil refinery and other chemical processes because it is suitable for high pressure application. Present work is a modification over the conventional shell and tube heat exchanger applying lumped tubes and helical baffles in conjunction with conventional STHXs. The modified design is compared with conventional STHXs by means of computational fluid dynamics (CFD) method Lumped tube will be widely used in chemical industry field. Pressure drop and heat transfer characteristics in the shell side are investigated with Realized ķ-ε model. Based on the periodic model, the STHX with discontinuous helical baffles (DCH-STHX) is investigated under the helix angle 22°.The results show that under the same mass flow rate M the shell side Nusselt number is 9.3-30.2% higher than that conventional STHXs. It is also reflected by the results that friction factor is increases with lumped tube in STHXs but in very less amount. The lumped tube and helical baffle STHXs gives better heat transfer performance than the conventional STHXs under the same mass flow rate. The 3D numerical simulation results also show that it gives more effective to improve the heat transfer performance of DCH-STHX, especially in the cases of large mass flow rate. Secondary flow is also reflected inside the tube and helix which are formed by lumped tube.

Keywords: Bell-Delaware Method, Nusselt number, Pressure drop, Heat transfer coefficient, Shell-and-tube heat exchangers, helical baffles

NOMENCLATURE

D_0 Tube outside diameter (m)

D_{otl} Tube outside limit (m)

D_s Shell inside diameter (m)

h_0 Shell side heat transfer coefficient (W/m^2-K)

h_{ideal} Ideal heat transfer coefficient in tube bundle

J_B Heat-transfer correction factor for tube bundle bypass

J_C Heat-transfer correction factor for baffle window flow

J_L Heat-transfer correction factor for baffle leakage effects

N_c Number of tubes row in between baffles

N_{cw} Effective number of tubes in one cycle

n_b Baffle number

P_t Pitch of tubes (m)

R_B Pressure drop correction factor

R_L Pressure drop correction factor for baffle leakage effect.

S_m Cross flow area (m^2)

Greek letters

ΔP_s Shell Side pressure Drop (Pa)

ΔP_c Cross flow Pressure drop (Pa)

μ Dynamic viscosity (Kg.s/m^2)

ρ Fluid density (Kg/m^3)

α Helix angle (degree)

Abbreviations

CAD Computer Aided Drafting

HTC Heat transfer coefficient

STHXs Shell and tube heat exchanger

Vs Versus

NA Not applicable

1.1 Overview

A heat exchanger is a device that is used to transfer thermal energy between two fluids at different temperatures and in thermal contact. It is equipment which transfers the energy from hot fluid to cold fluid with maximum rate and minimum investment and running cost. Heat exchangers are devices that facilitate the exchange of heat between two fluids that are at different temperature while keeping them from mixing with each other. The hot fluid gets cooled, and the cold fluid is heated. Heat exchangers are widely used for refining, electric power generation, chemical processing, and environment protection.

The design of Shell and Tube heat exchanger is mainly on the basis of two parameters which are operational and geometrical parameters like fluids temperature, mass flow rate, Flow arrangement, Material of STHXs, Tube length, Shell and Tube diameter, Baffle angle etc. Shell and tube heat exchanger (STHXs) are widely used in many industrial applications, such as thermal power plant, chemical engineering, petroleum refining, food industries etc. According to previous research it is found that more than 40% of Heat exchangers are Shell and tube heat exchanger due to their compact geometry and area density concept which is easy in maintenance and reduce fouling possibilities.

1.2 Flow Pattern in Heat Exchanger

Heat exchanger has three different types of heat exchanger:

1. **Parallel flow:** If hot fluid and cold fluid flows in the same direction, then hot fluids gets cooled and cold fluid gets hot. In parallel flow the inlet temperature difference is greater and this becomes lesser at the end of exchanger length at exit side. Between these two ends heat transfer takes place.
2. **Counter flow**: If hot and cold both flows in opposite direction to each other, then this arrangement is termed as counter flow. One in tube and another in shell side. A car radiator is good examples of cross flow heat exchangers. Cross flow heat exchangers are typically used for heat transfer between a gas and a liquid. In counter flow heat exchanger, maximum rate of heat transfer is achieves
3. **Cross flow**: - If hot fluid and cold fluid flow normal to each other, then this type of arrangements is called cross flow. Types of cross flow heat exchangers (a) Unmixed and (b) Mixed flow.

In shell and tube heat (STHXs) exchanger we use mainly parallel flow and counter flow. The temperature of the two fluids usually varies from inlet to exit of the heat exchanger, except in the case of phase change on either side when the temperature remains constant. The tube may be in the form of coil also. Generally we prefer the counter flow heat exchanger rather than parallel flow; because LMTD (logarithmic mean temperature difference) of counter flow heat exchanger is more than the parallel flow heat exchanger.

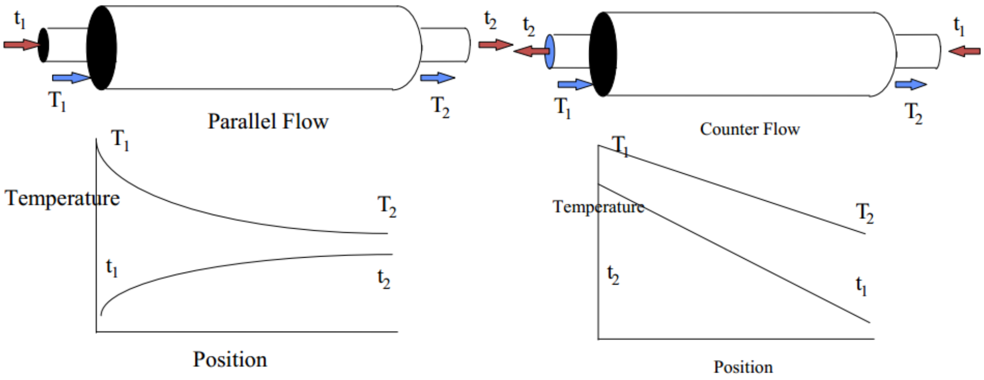

Figure 1.1 Parallel flow and Counter flow in heat exchanger.

1.3 Factors Affecting the Heat Exchanger

1.3.1 Surface area: To increase the heat transfer, heat exchangers are designed in such a way that leads to the maximum surface area of the wall between the two fluids, while providing resistance to fluid flow rushing without passing through tube bundles.

1.3.2 Fins: Performance of heat exchangers is also depends on fins or extended area for heat transfer, which increase its turbulence and greater mixing.

1.3.3 Baffle angles: Segmental baffles have some shortcomings and limitations on it, so to reduce this type of inherent deficiencies baffles inclined at some angle which reduces shell side pressure drop and back flows and provide helix flow throughout the shell and tube heat exchanger and ultimately heat transfer increased.

1.3.4 Geometry of tubes: Different geometry of tubes like coil pitches leads to greater heat transfer because it produces secondary flow which enhances its turbulence. The design of shell and tube heat exchanger is mainly depends on this parameter because it is the parameter which cause enhanced heat transfer.

1.3.5 Other factors: Some other factors must be encountered in heat exchanger like efficiency of heat exchanger, compactness, and heat transfer rate and pressure drop in shell side.

1.4 Shell and Tube Heat Exchanger

Shell and tube heat exchangers consist of a series of tubes. A large number of tubes are packed inside the shell which encloses all the tubes with the help of baffles. Baffles are mainly used to support the tubes bundle and to direct the flow in well manner on the other hand it is increase the heat transfer area and also increase contact time between fluids which gives us enhanced heat transfer. Tubes contain the fluid that must be either hot or cold the other fluid rushing over the tube bundle that are being hot or cold so that it can transfer the heat or absorb the heat. A large number of tubes in set are called the tube bundle and it can be different types like coiled tubes, twisted tube, helical tube finned tubes grooved tubes etc. Shell and tube heat exchangers are commonly used for high-pressure applications because its compactness features.

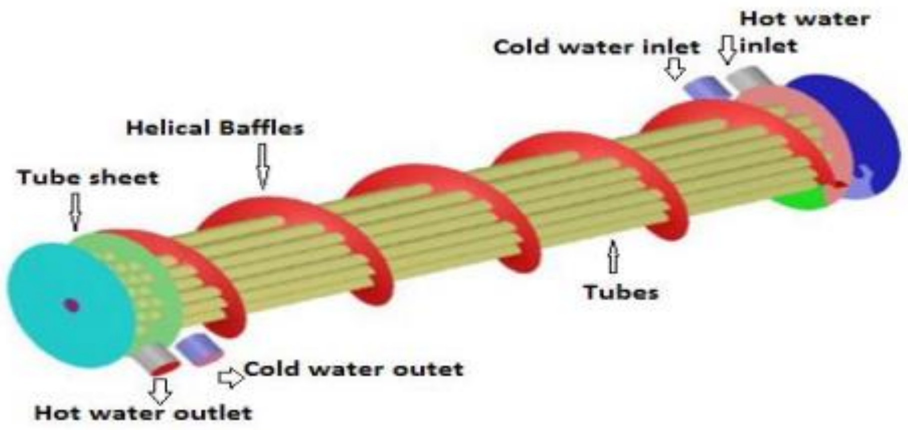

Figure 1.2 Schematic view of shell and tube heat exchanger (STHXs)

It is broadly used in industries purpose and power plants, nuclear reactor. Shell and tube heat exchanger is a very process device in high energy industry, it has sometimes different name in other application, such as the condenser, cooler and evaporator, but they are basically common in function. Unfortunately, low heat transfer efficiency caused by scaling the contamination is a major problem of shell and tube heat exchanger. Heat transfer enhancement is mainly realized by improving heat transfer surface area. Enhanced heat transfer of tube side is mainly realized by changing the shape of heat transfer surface or tubes inserts to enhance the fluid turbulence intensity and increase heat transfer area.

1.4.1 Basic Components of Shell and Tube Heat Exchanger

1) Tubes
2) Tube sheets.
3) Shell and shell side nozzles.
4) Tube side channels and nozzles.

5) Channel covers.

6) Pass divider.

7) Baffles.

1.4.2 Multiple Shell and Tube passes

These heat exchangers multiple shell and tube passes are used to enhance the heat transfer rate. Shell side fluid transverse more than once in exchanger by using multi pass. By proper designing the tube side fluid can travel back from one shell to another by external means in same way the shell side fluid can transverse more than once. Here shows the multiple passes shell and tube heat exchanger which are ultimately increase the contact time between the fluids and increase the heat transfer rate.

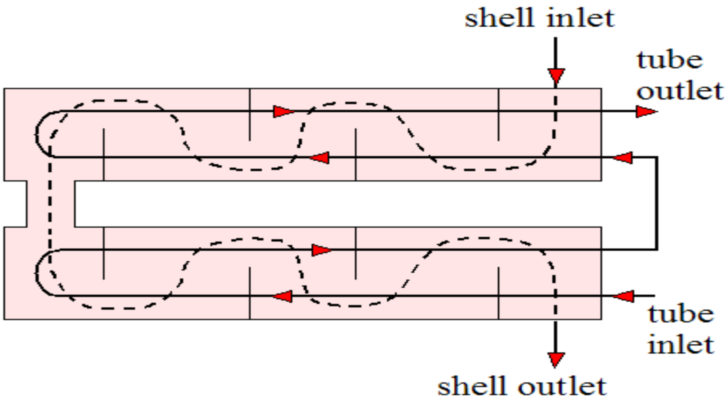

Figure 1.3 2 Shell and 4 Tube passes Heat exchanger

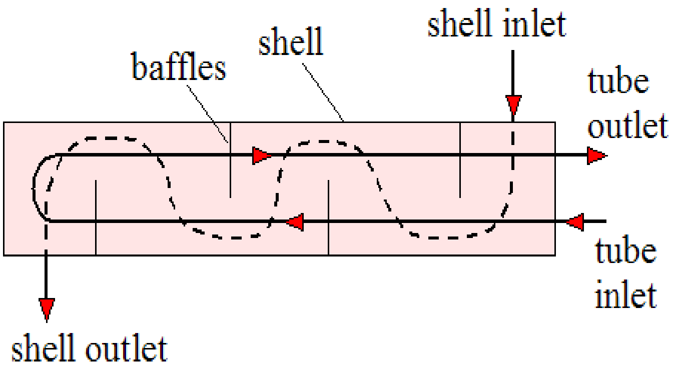

Figure 1.4 1Shell and 2 Tube passes Heat exchanger

1.5 Design Features of Shell and Tube Heat Exchanger

1.5.1 Diameter of tube

A small tube diameter is essential for heat exchanger both economical and compact. However, it is more likely for the heat exchanger to fouling faster and the small size tubes creates easiness in cleaning.

1.5.2 Thickness of tubes

The thickness of the tubes is determined to ensure that flow-induced vibration has resistance. It is also tells about its axial strength that how much load it can sustain. It is also ensuring about its hoop strength (to withstand internal tube pressure) and buckling strength (to withstand overpressure in the shell).

1.5.3 Corrugation in Tubes:

Corrugation in tube is mainly used for the inside the tubes, turbulence and greater mixing of the fluids is increases by the corrugation which gives better heat transfer performance.

1.5.4 Layout of Tubes

There are four types of tube layout, which are square and rotated square triangular rotated triangular , The triangular shaped are used to give greater heat transfer as they force the fluid to flow in a more turbulent around the piping.

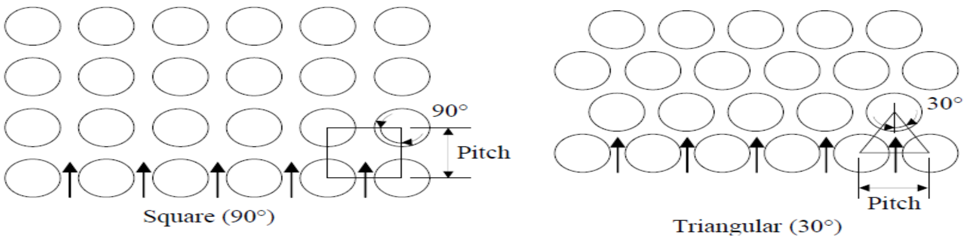

Figure 1.5 Tubes layout

1.5.5 Length of Tubes

More preferable tubes when it has smaller shell diameter and a long tube length. It has some limitations over space available and it is ensure that tubes are available in lengths also, it is found long thin tubes are difficult to cleaning and take out and replace.

1.5.6 Design of Baffles

Baffles are mainly used in shell and tube heat exchangers to give direction to fluid across the tube bundle. Baffles are mainly used to support the tubes bundle and to direct the flow in well manner on the other hand it is increase the heat transfer area and also increase contact time between fluids which gives us enhanced heat transfer. The most common type of baffle is the segmental or conventional baffle.

1.6 Classification of STHX based on service

Basically, a service may be single-phase (such as the cooling or heating of a liquid or gas) or two-phase (such as condensing or vaporizing). Since there are two sides to an STHE, this can lead to several combinations of services. Broadly, services can be classified as follows:

1.Single-phase (both shell side and tube side);

2.condensing (one side condensing and the other single-phase)

3.vaporizing (one side vaporizing and the other side single-phase)
4.Condensing/vaporizing (one side condensing and the other side vaporizing).

The following nomenclature is usually used

Heat exchanger: both sides single-phase and process streams (that is, not a utility).

Cooler: one streams a process fluid and the other cooling water or air.

Heater: one streams a process fluid and the other a hot utility, such as steam or hot oil.

Condenser: one streams a condensing vapor and the other cooling water or air.

Chiller: one streams a process fluid being condensed at sub-atmospheric temperatures and the other a boiling refrigerant or process stream.

Reboiler: one stream a bottoms stream from a distillation column and the other a hot utility (steam or hot oil) or a process stream.

1.7 Designing parameter of shell and tube heat exchanger

1.7.1 Flow rates: flow rate of both streams must be consider

1.7.2 Inlet and outlet temperatures: Temperature is an important parameter in designing the HE both inlet and outlet.

1.7.3 Operating pressure of both streams, this is required for gases, especially if the gas density is not furnished, it is not really necessary for liquids, as their properties do not vary with pressure.

1.7.4Allowable pressure drop for both streams. This is a very important parameter for heat exchanger design. Generally, for liquids, a value of 0.5–0.7 kg/cm2is permitted per shell. A higher pressure drop is usually war-ranted for viscous liquids, especially in the tube side. For gases, the allowed value is generally 0.05–0.2 kg/cm2, with 0.1 kg/cm2 being typical.

1.7.5 Fouling resistance for both streams. If this is not furnished, the designer should adopt values specified in the TEMA standards or based on past experience.

1.7.6 Physical properties of both streams. These include viscosity, thermal conductivity, density, and specific heat, preferably at both inlet and outlet temperatures. Viscosity data must be supplied at inlet and outlet temperatures, especially for liquids, since the variation with temperature may be considerable and is irregular (neither linear nor log-log).

1.7.7 Heat duty the duty specified should be consistent for both the shell side and the tube side.

1.7.8 Type of heat exchanger if not furnished, the designer can choose this based upon the characteristics of the various types of construction de-scribed earlier. In fact, the designer is normally in a better position than the process engineer to do this.

1.7.9 Line sizes it is desirable to match nozzle sizes with line sizes to avoid expanders or reducers. However, sizing criteria for nozzles are usually more stringent than for lines, especially for the shell side inlet. Consequently, nozzle sizes must sometimes be one size (or even more in exceptional circumstances) larger than the corresponding line sizes, especially for small lines.

1.7.10 Preferred tube size Tube size is designated as O.D. ×thickness × length. Some plant owners have a preferred O.D. ×thickness (usually based upon inventory considerations), and the available plot area will deter-mine the maximum tube length. Many plant owners prefer to standardize all three dimensions, again based upon inventory considerations.

1.7.11 Maximum shell diameter this is based upon tube-bundle removal requirements and is limited by crane capacities. Such limitations apply only to exchangers with removable tube bundles, namely U-tube and floating-head. For fixed-tube sheet exchangers, the only limitation is the manufacturer's fabrication capability and the avail-ability of components such as dished ends and flanges. Thus, floating-head heat exchangers are often limited to a shell I.D. of 1.4–1.5 m and a tube length of 6 m or 9 m, whereas fixed-tube sheet heat exchangers can have shells as large as 3 m and tubes lengths up to 12 m or more.

1.7.12 Materials of construction if the tubes and shell are made of identical materials, all components should be of this material. Thus, only the shell and tube materials of construction need to be specified. However, if the shell and tubes are of different metallurgy, the materials of all principal components should be specified to avoid any ambiguity. The principal components are shell (and shell cover), tubes, channel (and channel cover), tube sheets, and baffles. Tube sheets may be lined or clad.

1.7.13 Special considerations these include cycling, upset conditions, alternative operating scenarios, and whether operation is continuous or intermittent.

1.8 Helical baffle Shell and Lumped Tube Heat Exchanger:

This type of heat exchanger is shown very efficiently performance especially for the cases in which the heat transfer coefficient in shell side is able to controlled or less pressure drop achieved. Circular half shaped baffle segments are arranged at an angle to the tube axis in a sequential pattern that helps the shell side fluid to flow in a helical manner over the tube bundle.

The tube is in lumped shaped that leads to increase the surface area for heat transfer without attaching the fin and it also leads to create turbulence flow inside the tube that cause better mixing between the fluids that is ultimately increase its heat transfer coefficient. This combination will definitely increase the rate of heat transfer and provide fluid to flow in helical path of shell side throughout the length of Shell and tube heat exchanger.

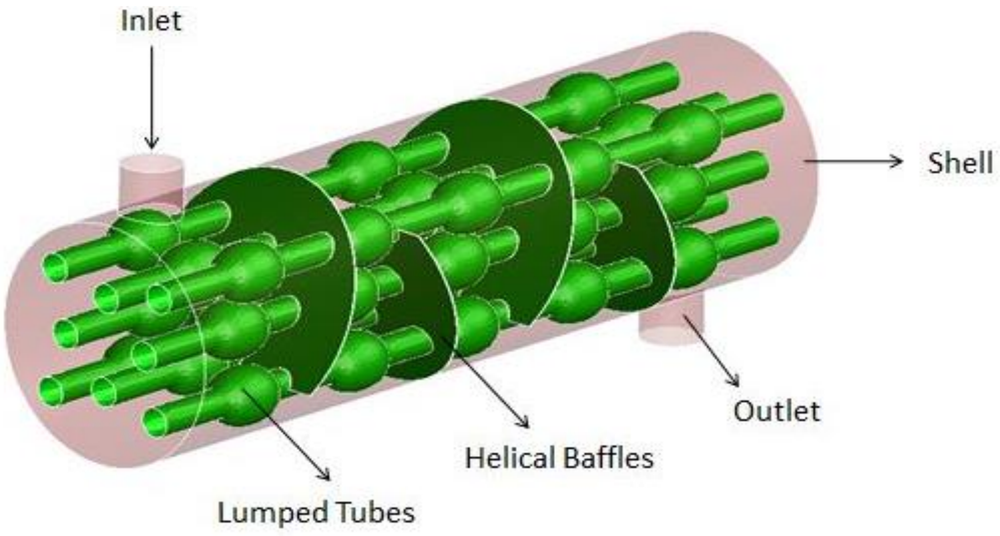

Figure 1.6 Helical baffled shell and Lumped tube Heat exchanger**.**

The concept of lumped tube and helical baffle shell and tube heat exchanger helps to remove many of inherent deficiencies of conventional heat exchanger. The tube is in lumped shaped that leads to increase the surface area for heat transfer without attaching the fin and it also leads to create turbulent flow inside the tube that cause better mixing between the fluids that ultimately increase its heat transfer coefficient. This combination will definitely increase the rate of heat transfer and provide fluid to flow in helical path of shell side throughout the length of Shell and tube heat exchanger. The performance of this heat exchanger will depends on helix angle which determines the pressure drop on shell side i.e. pumping power required. The heat transfer per unit pressure drop is good metric for comparing the performance.

In order to overcome the difficulty in manufacturing point of view, the discontinuous helical baffle is used; however leakage in discontinuous helical baffle is relatively large due to the triangular dead zones, results in reduced heat transfer performance. To overcome this limitation researcher introduced continuous helical baffle and found superior performance in heat transfer over conventional shell and tube heat exchanger. But the manufacturing of continuous helical baffle is complicated than that of discontinuous helical baffle. Thus, in order to simplify the manufacturing problem and make complete use of helical baffles shell and lumped tube is introduced in shell and tube heat exchanger. This modified design reduces cross- sectional flow area, velocity of fluid increases and can have great improvement in heat transfer. In addition it will reduce manufacturing difficulty and lower pressure drop.

Computational Fluid Dynamics (CFD): As a research tool it is used to solve complex flow and heat transfer problems. The use of CFD is to predict the internal and external flow. It is a high level of operation requires a high level of skill and understanding from the operator to obtain meaningful results in complex situation.

Heat Exchanger: It is equipment which is built for efficient transfer of heat between two mediums. Heat exchanger is a device that facilitates the exchange of heat between two fluids at different temperature .Heat transfer media can be a direct contact type or can be separated by a wall. Heat exchangers are widely used in refrigeration & air conditioning, power plants, petroleum refiners, natural gas processing and in many more applications. A heat exchanger categorizes in general three types, namely:

1. Transfer type HE.
2. Storage type HE.
3. Direct contact type HE.

Transfer type HE: In this type of HE fluids are kept separate and they do not mix as the flow through it. Heat is transferred through the separating walls. A concentric double pipe heat exchanger is an example of this type. Present study is based on analysis of transfer type heat exchanger.

Shell-and-Tube HE: It consists tubes in series. The heated or cooled water flows through one set of pipe. The second fluid runs in other set of tubes which are being cooled or heated, so that first set of tube can absorb or transfer the heat to second set of tubes according to requirements.

There are some technical terms which are used in present study as follows:

Reynolds number (Re): Reynolds number of airflow is calculated by knowing velocity, density, hydraulic diameter of duct and viscosity. Reynolds number is important parameter which decides the turbulence of fluid through duct. If Re ≥2300 then flow is turbulent, otherwise it is laminar flow. Re is calculated by

$$Re = \frac{\rho V D_h}{\mu} \tag{2.1}$$

Where D_h is the hydraulic diameter and calculated by

$$D_h = \frac{4A_C}{P} \tag{2.2}$$

Nusselt Number: It is a dimensionless form of heat transfer coefficient which is the ratio of rate of heat transfer through convection to the rate of heat transfer through conduction.

$$Nu = \frac{hL}{K} \tag{2.3}$$

Heat transfer rate (Q): It is the phenomena where hot fluid gets cooled and cold fluids gets hot means heat is gain by cold fluid and heat is lost bye hot fluid by this phenomena they both are in thermal equilibrium.

Heat loss by hot water

$$\mathrm{Q_h = m_h \times C_{ph} \times (T_{hi} - T_{ho})} \tag{2.4}$$

Heat gained by cold water

$$\mathrm{Q_C = m_c \times C_{pc} \times (T_{ci} - T_{co})} \tag{2.5}$$

Where, Q = Q_c = Q_h (kw)

Logarithmic mean temperature difference (LMTD): It is defined as average driving temperature difference for heat transfer between two fluids flowing at different temperature. Given as

$$\mathrm{LMTD} = \frac{\Delta T_1 - \Delta T_2}{\ln(\Delta T_1/\Delta T_2)} \tag{2.6}$$

Where $\Delta T_1 = T_{hi} - T_{co}$ & $\Delta T_2 = T_{ho} - T_{ci}$

Effectiveness (ϵ): It's a ratio of actual heat transfer to the maximum heat transfer. The maximum possible heat transfer depends on one of the fluids undergoing the maximum possible change in temperature and that will be the fluid which will have the minimum value of heat capacity rate.

Given as below

$$\epsilon = \frac{Q_a}{Q_{max}} \tag{2.7}$$

Overall Heat Transfer Coefficient (U): Heat exchanger is designed on the basis of lower heat transfer coefficient. It can used to calculate the total heat transfer through heat exchangers or walls. Given as

$$\mathrm{U} = \frac{\mathrm{Q}}{A_s \times \Delta T_{ln}} \tag{2.8}$$

Fouling (R_f): The performance of heat exchangers usually reduces as the time passing as a result there is accumulation of deposits on surfaces. This layer of deposits creates resistance on the path of heat transfer and reduces heat transfer capacity.

Tube Clearance (C): It is distance between two successive tubes which is given by

$$C = P_t - D_{out} \tag{2.9}$$

Where, P_t =Tube pitch, D_{out} = Outer tube diameter.

Cross Flow area: The cross flow area along the centerline of flow in shell given as

$$S_m = D[(D_s - D_{out}) + \frac{(D_{out} - D_0)}{P_t} \tag{2.10}$$

Where D_s = Shell diameter,

Maximum Velocity: Shell side velocity which is depends upon cross flow area is given by

$$V_{max} = \frac{Q_s}{A}\ (m/s) \tag{2.11}$$

Pressure Drop (ΔP): It signifies the drop in pressure due to fiction losses in heat exchanger. It is directly leads to pumping cost of mechanical equipment given as

$$\Delta P_s = [(n_b - 1) \times \Delta P_c \times R_B + n_b . \Delta P_w] R_L + 2\Delta P_c \left(1 + {}^{N_{cw}}/_{N_c}\right) R_B \tag{2.12}$$

Heat Capacity: The fluid with a large heat capacity rate will experience a small temperature change, and the fluid with a small heat capacity rate will experience a large temperature change. Heat capacity rate is given by

$$C_h = m_h C_{ph} \quad \& \quad C_c = m_c C_{pc} \tag{2.13}$$

From the survey related to shell and tube heat exchanger it has been observed that the geometry parameters also affect the Heat Exchanger performance.

It is quite difficult to manufacture continuous helical baffles. So there is a scope to work on heat exchanger with discontinuous helical baffles it is easy to manufacture as compare to helical baffle with lumped tube for giving the same heat performance as continuous helical baffles give. Also there was no work found in literature survey, based on the effort of lumped tube with helical baffle.

There is various changes is done in design of shell and tube heat exchanger

1. They used spiral type of tube to enhance the heat transfer but that geometry is very difficult to manufacture as compare to conventional heat exchanger.
2. They used twisted tube in shell and tube heat exchanger, also in this we are facing same type of problem as above mentioned.
3. They used helical pitched coil tube to optimize the heat exchanger design, but there is more fouling in heat exchanger and it is mandatory to clean the tube after some time, and those points of view this type of tube face draw back.
4. Some researcher used grooved tube that's directly enhance the heat transfer , but that also leads to cost of manufacture more as compare to conventional shell and tube heat exchanger.

So there is scope of study which holds easy for both the purpose enhanced heat transfer as well as less manufacture cost as compare to previous researched have done. This is also easy for cleaning so less fouling is there. So for completing this project we have to do -

1. To determine by simulation heat transfer & pressure drop in shell and lumped tube heat exchanger by using CFD-Fluent software.
2. To validate the results this is obtained by Fluent using mathematical modeling.

1. To enhance the heat transfer rate by the optimization design of shell and tube heat exchanger.
2. To develop a CFD simulation to predict heat transfer & pressure drop in helical baffled shell and lumped tube.
3. To increase contact time and contact surface area of fluids with each other which gives best integrated performance in terms of heat transfer and pressure drop.
4. Thermal analysis of vibration reduced design shell and tube heat exchanger.
5. Increase the effectiveness of shell and tube heat exchanger.
6. Determining pressure, velocity and temperature profiles for heat flux which I will obtain after analysis.
7. Determining temperature distribution along the length of the shell and tube heat exchanger.
8. To determine average heat transfer coefficient and Nusselt number for different mass flow rate.

Jian Feng Yang et al. [8] investigated the effects of sealing strips on shell-side flow and heat transfer performance of heat exchanger with helical baffles, numerical simulations are carried out to investigate the effects of number and width of the sealing strips on shell-side flow and heat transfer of a shell-and-tube heat exchanger (STHX) with helical baffles. The STHX with discontinuous helical baffles (DCH-STHX) and the STHX with continuous helical baffles (CH-STHX) are investigated under the same helix angle 40°which is the optimal angle proved by the previous research studies. The results shows that with the increase of the number of sealing strip, the shell side Nusselt number is 9.3-41.7% higher than that without sealing strips while the resistance increases by 37.5-189.7%.

Farhad et al. [4] analytically studied they used helical baffles with 40° helix angle with different baffle spacing. Baffle spacing is differentiated on the basis of pitch P. There are five cases with different baffle spaces 15mm, P/16, P/8, 3P/16, and P/4 respectively. The results shows that for the same mass flow rate heat transfer coefficient per unit pressure drop increases with decrease in baffle space. It is also found that for the same mass flow rate with the increasing of baffle space heat transfer coefficient decreases. It is also observed that pressure gradient decreases as we move away from the sequence.

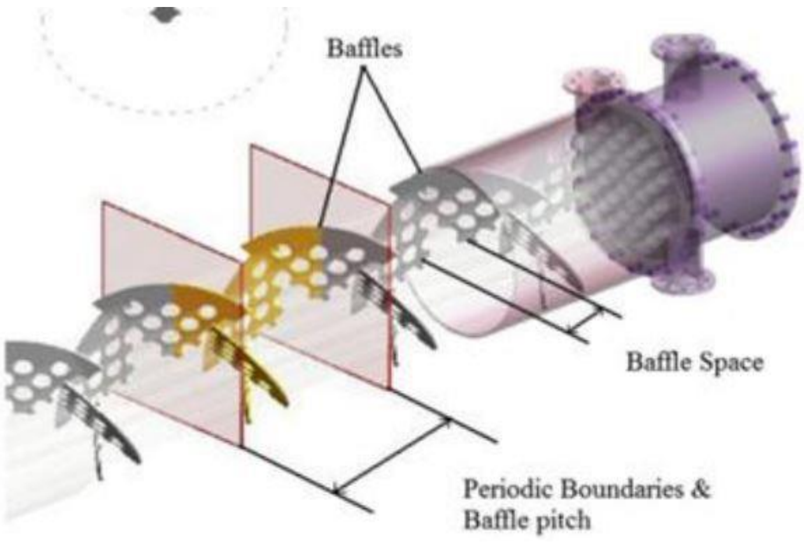

Figure 5.1 Impact of baffle space in STHXs

Zhang et al. [10] helical baffles generated a flow patterns are close to plug flow which leads reduction in shell side pressure drop and enhance the heat transfer performance. They studied on different baffles angle with discontinuous helical baffle 20°, 30°, 40°, 50°. It is found that the heat transfer coefficient decrease with increase in helix angle. It is also found that pressure drop increases with increasing mass flow rate due to decrement in shell side velocity. Helical baffles generated a flow patterns are close to plug flow which leads reduction in shell side pressure drop and enhance the heat transfer performance.

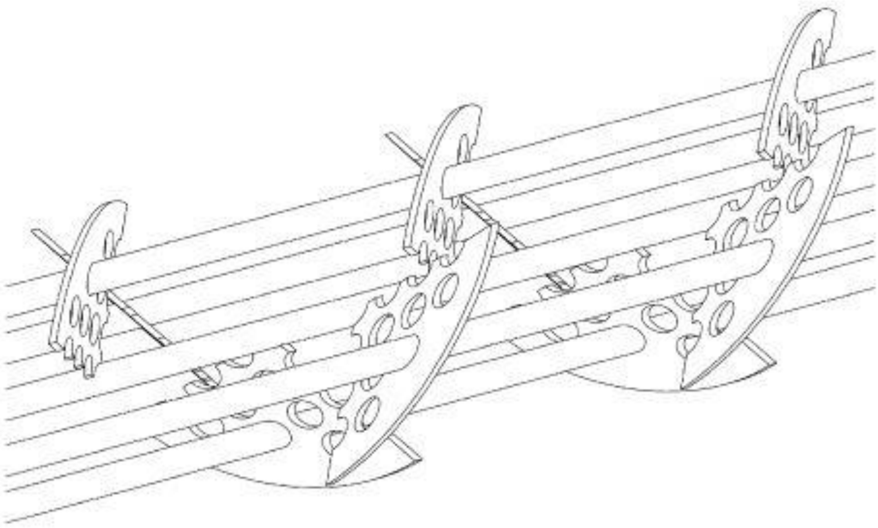

Figure 5.2 Overlapped helical baffle arrangement.

Shinde et al. [18] comparatively studied on shell and tube heat exchanger with conventional STHXs with segmental baffles and modified STHXs with helical baffles using Bell-Delaware method and they found that ratio of heat transfer coefficient per unit pressure drop is extremely high in case of helical baffle STHXs due to elimination of shortcomings in conventional STHXs.

Shinde et al. [19] have been also studied on Shell side performance of Helix changer. From the analysis 40° is the optimum angle where it gives high heat transfer coefficient per unit pressure drop. They studied on continuous baffle and found that 25° is optimum angle for the helix flow and produce better heat transfer coefficient per unit pressure drop.

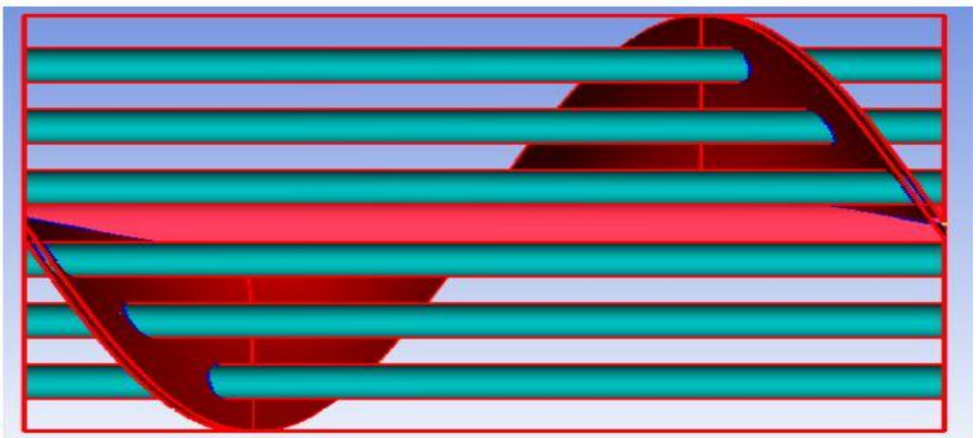

Figure 5.3 Helixchanger with center tube.

Salimpour [12] studied on shell and coiled tube heat exchanger taking into consideration coiled tube of both parallel flow and counter flow and they proposed a correlation for shell side and tube side. Coiled tube leads to secondary flow inside the tube which causes better missing between the fluids.

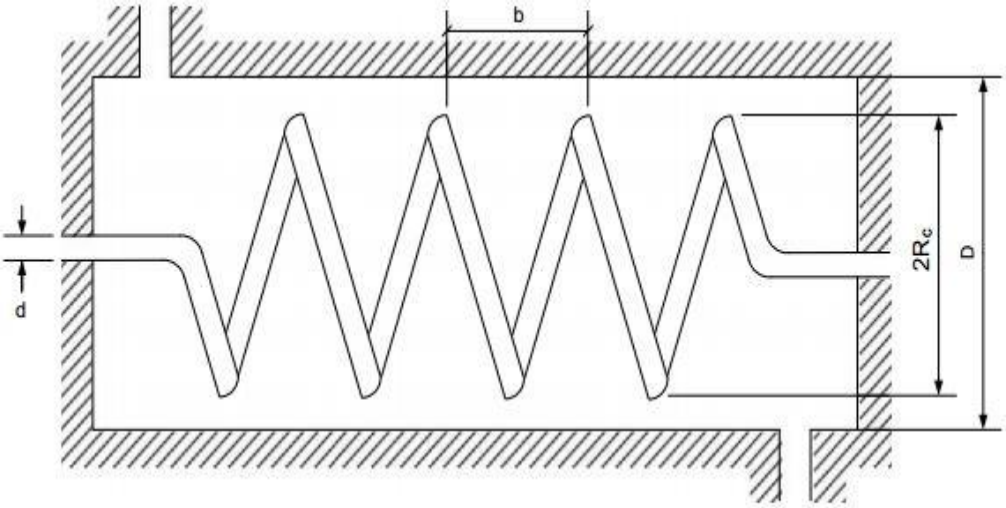

Figure 5.4 Coiled pitch tube Heat exchanger

Xiang-hui Tan, et al. [23] investigated performances of twisted oval tube heat exchanger. Results shows that Nusselt number and friction factor both increase with the increasing of P and A/B and it is also concluded that heat transfer performance on shell side increases with increase in A/B ratio but on the aspects on P it is firstly increase with increasing in P after that it will start decreasing.

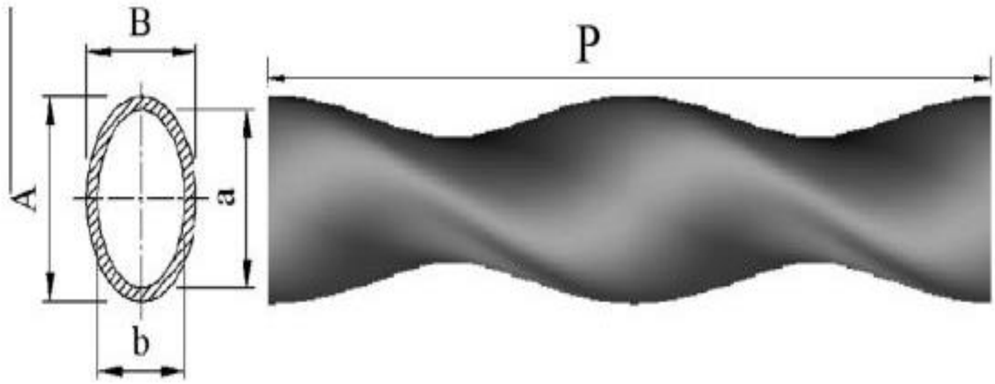

Figure 5.5 Sketch of twisted oval tube.

Choon et al. [22] analytically studied shell and double concentric tube heat exchanger. In these sixty six tubes is enclosed with one shell. This is carried by computational fluid dynamics and it has three inlets and three outlets. This design will lead to increase in design complexity and of Corse manufacturing cost and cover lot of space. It is found by the comparison with the Kern Method; it is a good agreement with each other and has acceptable percentage errors.

Bolinder [1] expressed governing equation with tensor analysis method, studied on curved pipes having variable cross-section like helical duct and rotating helical duct and it is found that secondary flow in twisted tube. Because of secondary flow there will be turbulences in pipe wall contact which leads to greater mixing between the fluids.

Qiuwang Wang et al. [17]. The results show that the pressure loss reduces in CMSP-STHX for the same mass flow rate heat transfer coefficient increases. The combined shell and tube is compared with the conventional shell-and-tube heat exchanger with segmental baffles (SG - STHX) by means of computational fluid dynamics (CFD) method. And it is found that combined shell and tube heat exchanger has greater performance over segmental baffle shell and tube heat exchanger. It is found that for the same pressure drop the heat transfer coefficient is 5.5% higher in CMSP-STHXs than the SG-STHXs.

Jahanmir et al. [6] Studied twisted bundle Heat exchanger using CFD tool, shell and tube heat exchanger examined under single twisted tube with five different twist angles. It is found that heat transfer coefficient is smaller than the conventional STHXs while the pressure drop is much smaller in twisted tube bundle. Optimum bundle twist angles for such exchangers are found to be 65 and 55° for all shell side flow rates.

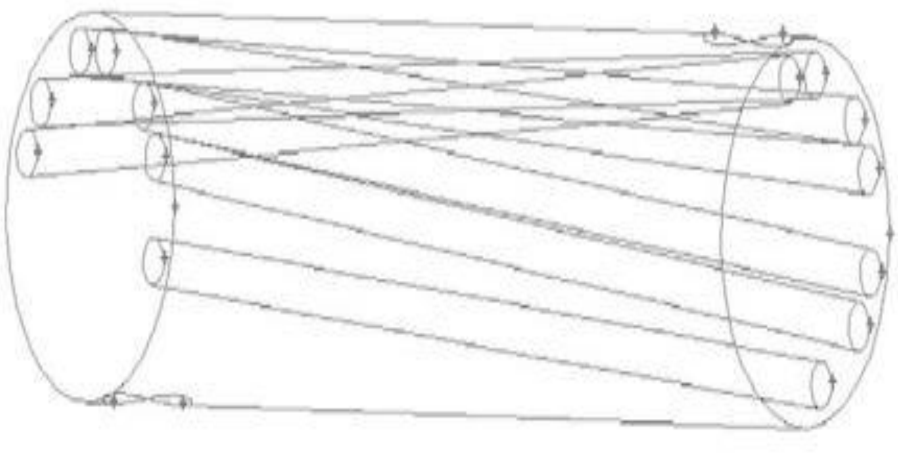

Figure 5.6 Twisted tube bundles with twist angle of 55°.

Jian Wen, et al. [8] studied on sealers to block the bypass which ultimately enhance the heat transfer coefficient which is increased up to 18.2-25.5%. Pressure losses increased by 44.6–48.8% with the sealer, but the increment of required pump power can be neglected compared with the increment of heat transfer coefficient. At the end it is concluded that heat transfer coefficient per unit pressure drop is ultimately increases.

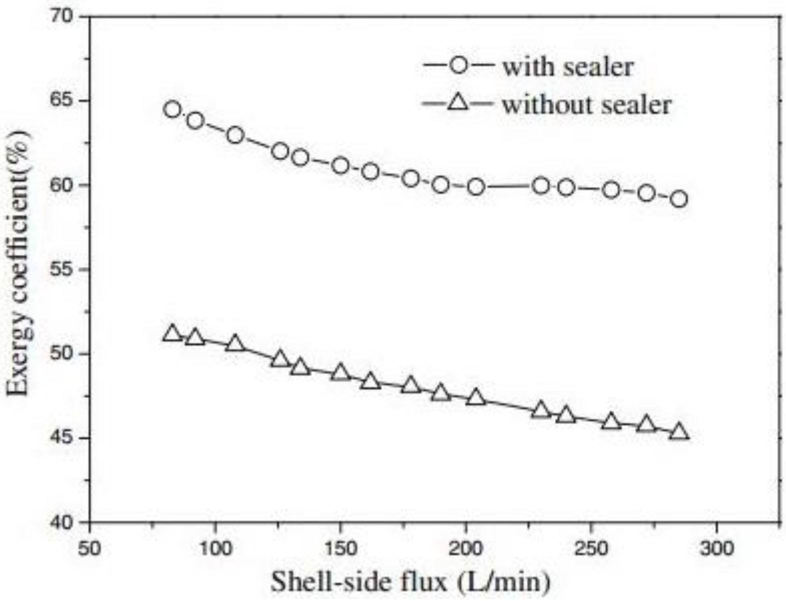

Figure 5.7 Comparative Performance of STHXs.

Lin Guang-yi et al. [11] experimented in heat exchanger inserted with rotors. In this they employed a rotor inside the tube for greater efficiency and create more turbulence inside the tubes. They inserted rotor specification with 19-100 and 19-400 in tube and it is observed that tube inserted with 19-400 gives better performance. It shows greater heat transfer coefficient when flow velocity is become greater than critical velocity.

Jie Yang et al. [9] have been studied a comparison of four numerical modeling approaches for enhanced shell-and-tube heat exchangers with experimental validation. They studied four different modeling approaches of a rod-baffle heat exchanger, which are the unit model, the periodic model, the porous model, and the whole model. The experiments validate the precision of each model in predicting heat transfer and pressure drop with the experimental validation. It is concluded that the periodic model, porous model and whole model have a high accuracy on predicting heat transfer performance, while the unit model has a relatively low accuracy. It is concluded that the porous model and whole model has high accuracy on predicting pressure drop, while unit model and periodic model are unable to directly predict hydraulic performance. The porous model with relatively high precision consumes medium numerical resources. However, it requires extra codes and is not applicable for new designs; the periodic model consumes relatively low resources and the unit model consumes the lowest.

Mohammadi [14] investigated the effect of baffle orientation on the performance of heat exchanger. He used vertical and horizontal orientation of baffles for testing in CFD. He found that. The shell side pressure drop and average Nusselt no. are calculated for different Reynolds no. The shell side pressure drop and heat transfer for vertical baffle heat exchanger was greater than horizontal baffle.

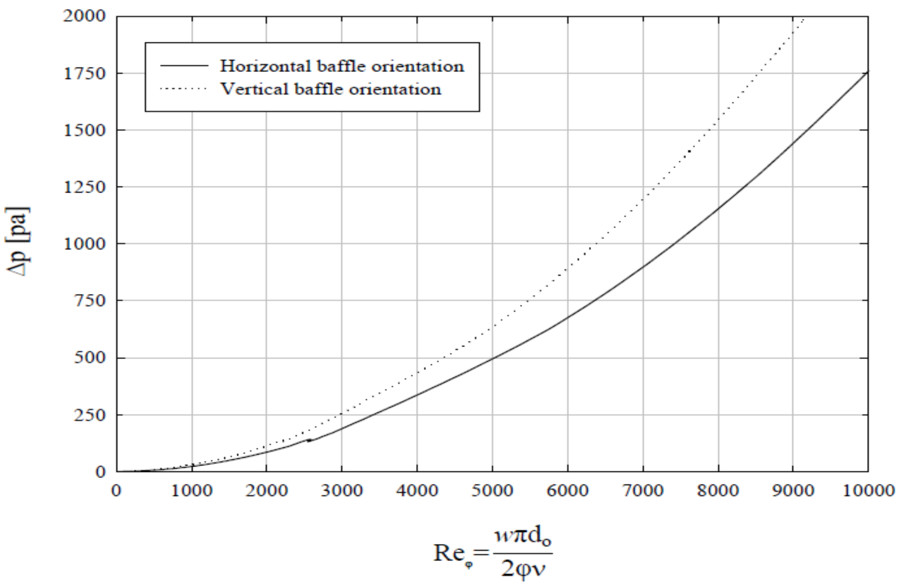

Figure 5.8 Pressure drop Vs. Reynolds no.

Gopi Chand et al. [5] investigated the thermal performance of water and oil type shell and tube heat exchangers. This type of heat exchanger mostly used in oil coolers, pre heaters and condensers etc. The high pressure can be done in shell and tube heat exchanger due to its robustness and medium weight. They have done the thermal analysis of these heat exchangers by using theoretical formulae. By using Floefd software they did the thermal analysis of water and oil type shell and tube heat exchanger. Theoretical results are compared with predicted results, which are calculated by Floefd software. They found a very good agreement between these results.

Mao-Yu Wen et al. [13] have been studied the heat transfer enhancement in fin and tube heat exchanger with improved fin design. In a wind tunnel three different types of fins they were used. They analyzed hew the drop of pressure for air side (), heat transfer coefficient (h), fanning friction factor (f) and the Colburn factor (j) get affected by air velocity (V) and Reynolds number (Re). They found that wavy fin as compare to flat fin show heat transfer coefficient (h), pressure drop, Colburn factor and friction factor are increases about 11.8-24.0%, 10.9–31.9%, 0.5–2.7% and 2.2–26.5% respectively. The compounded fin as compare to flat fin show heat transfer coefficient (h), pressure drop, Colburn factor and friction factor are increases about 28.0– 45.5%, 33.6–63.1%, 9.4–13.2% and 6.9–71.1% respectively. They suggest using the heat exchanger having compounded fins.

Ender Ozden et al. [3] have been done the investigation of small shell and tube heat exchange by using CFD software. They found how the heat transfer coefficient and the pressure drop depend on shell diameter, baffle cut and baffle spacing dependence. Numerical modeling also has been done by them. The shell and single tube pass heat exchanger, which is having variable number of baffles, are analyzed by CFD software for turbulent flow. The results are validated by the comparison of outlet temperature, heat transfer coefficient and pressure drop for their model with two baffle cut values. They discussed how the heat exchanger performance affected by the baffle spacing to shell diameter ratio at varying mass flow rate. The results are shown in figure below:

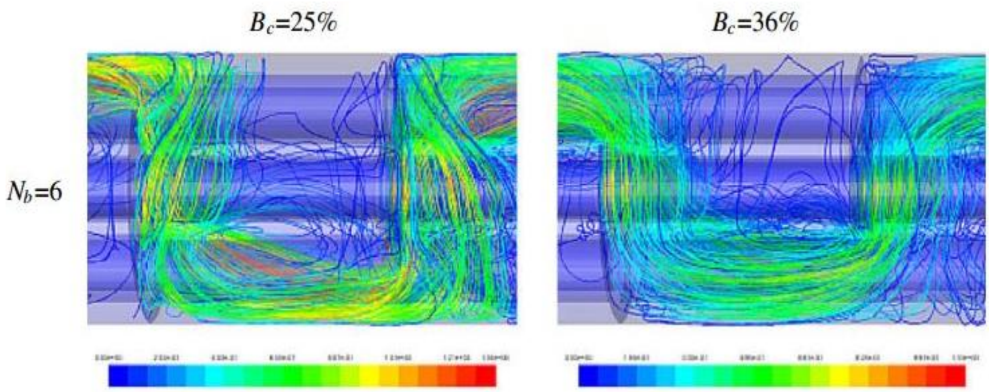

Figure 5.9 CFD analyses of different baffle cut and baffles.

Pravin et.al [15] studied on the physical importance of helix baffle in heat exchanger and what should be the optimum angle for the helix changer. They tested helix changer over segmental heat exchanger and found that running time foe helical baffle is 3 times more than segmental baffle. It is clear from picture that fouling is more in segmental baffle than helical baffle.

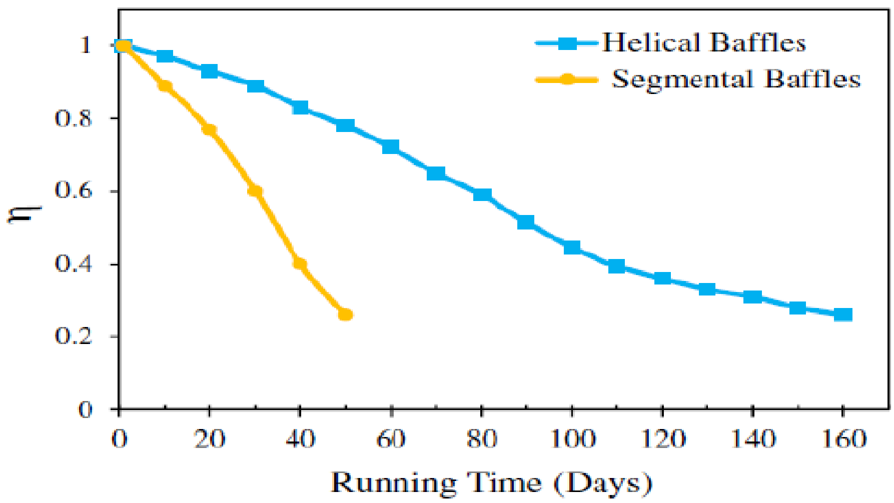

Figure 5.10 Running time of helical and segmental baffle.

He compared results of heat transfer and pressure drop for different helix angle and gives the conclusion that for lower helix angle heat transfer coefficient will be more. He got the heat transfer to pressure drop maximum for helix angle >35 degree.

Chen G.D.et al. [2] studied on periodic model of shell and tube heat exchanger. In the periodic models, the inlet and outlet faces set as periodic boundary conditions, tube walls were set as solid wall face conditions with constant temperature and shell face and baffle surface were set as wall-face conditions. The assumed upstream bulk temperature of the hot oil in the shell side is 335 K and the heat exchange tubes wall temperatures are the mean temperature on the inlet and outlet in the tube side measured in the experiments.

Entire research process will define in research methodology. There are several steps which should be encounter for the completion of the project.

Steps: 1 Define Problem

Thermal performance analysis of passive techniques of helical baffled shell and lumped tube heat exchanger. Analysis of heat transfer coefficient per unit pressure drop is calculated.

Steps: 2 Literature Survey

1. Survey review of past research and papers such as earlier work and articles related to shell and tube heat exchanger.
2. Review of heat transfer phenomena in shell and tube heat exchanger.
3. What are the main factors which directly affect the pumping cost.

Steps: 3 Design & Modeling

A conventional shell and tube heat exchangers, while having a good record of functionality in industries application, shows some shortcomings and limitations. In conventional heat exchanger fluid is not properly contact with tube bundle that may cause lower heat transfer and shows comparatively higher pressure drop. A modified shell and tube heat exchanger is shown in figure 1. It consists of helical baffle with helix angle (α) 22° and lumped tube instead of normal tube. The dimensions of the modified shell and lumped tube heat exchanger are as are:

Table 6.1 Geometrical parameters of shell and lumped tube heat exchanger

Sr. No	Geometrical parameters	Dimensions (mm)
1	Shell inner diameter	205
2	Shell outer diameter	225
3	Tube inner diameter	25
4	Tube outer diameter	26
5	Tube length	700
6	Total number of tubes	8
7	Tube arrangement	circular
8	Tube pitch	56.56
9	Tube pass	1
10	Shell pass	1
11	Number of Baffle	4
12	Baffle spacing	125

13	Helix angle	22°
14	Baffle cut	25

6.1 Geometry-CAD model:

1. Firstly modeled is designed in such way that it could be help to enhance the heat transfer between the two fluids.
2. Modeled is designed by using solid works software.

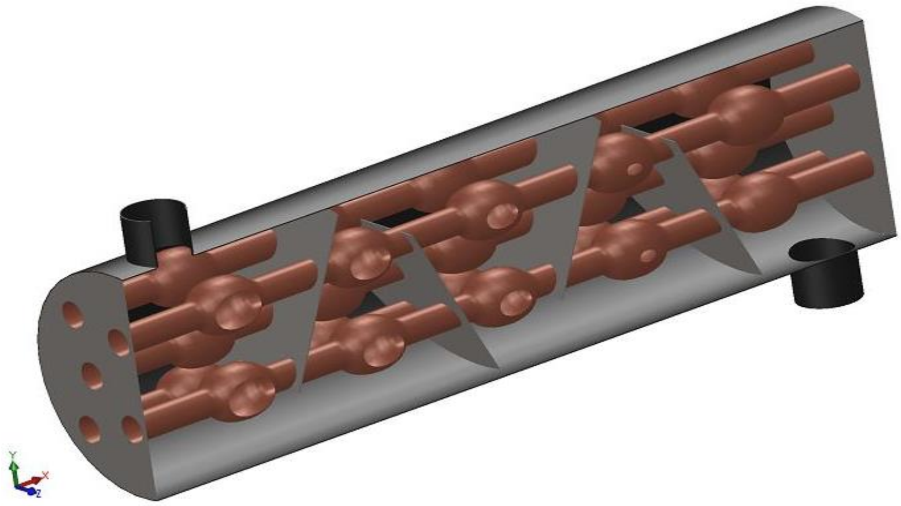

Figure 6.1 Helical baffled shell and lumped tube heat exchanger.

Steps: 4 Analysis using CFD

6.2 Computational Fluid Dynamics (CFD)

The use of Computational Fluid Dynamics (CFD) to predict internal and external flow has risen dramatically in the past decade. CFD operation requires high level of skill and understanding from the operator to obtain meaningful result in complex situations. Computational Fluid dynamic (CFD) is the science of predicting fluid low, Heat transfer, mass transfer, chemical reactions and related phenomena.

The result of CFD analysis is relevant engineering data used in conceptual studies of new designs, detailed product development. The advantages of simulating model in CFD rather than experimentation it, are as follows:

1. Reduction time taken and cost of new design.
2. Able to study the system where controlled experiments are difficult or impossible to perform, e.g. very large system.
3. Able to study system under hazardous conditions at and beyond the normal performance limits, e.g. safety studies & accident scenarios.

6.3 Working of CFD code:

CFD works mainly on three elements:

1. Pre-processor
2. Solver
3. Post processor

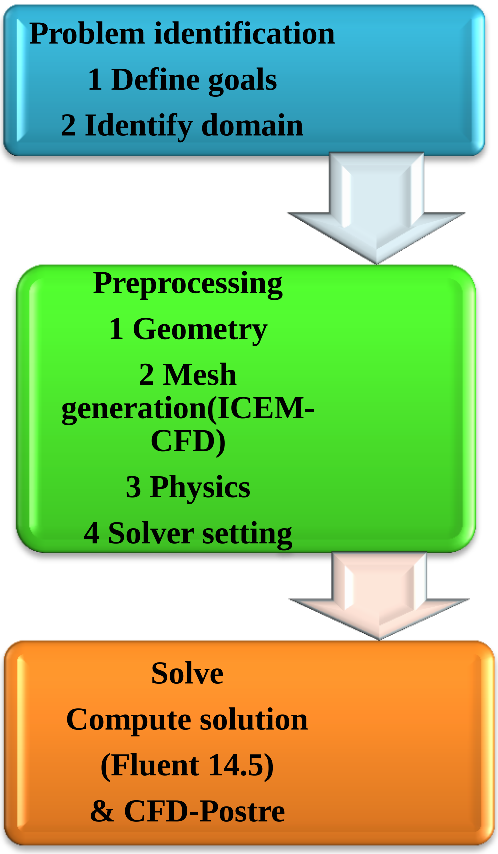

Figure 6.2 Flow chart of whole analysis process in CFD

1. Pre-processor:

Pre-processor consist of input of a flow problem to a CFD program by means of an operator friendly interface and the subsequent transformation of this input into a form suitable for use by the solver.

The pre-processor works in stages as follows:

1. Definition of the geometry of the region of interest: the computational domain.
2. Grid generation i.e. the sub-division of the domain into a number of smaller, non - overlapping sub-domains: a grid (or mesh) of cells (or control volume or elements).
3. Selection of the physical and chemical phenomena that need to be modeled.
4. Definition of fluid properties.

5. Specification of appropriate boundary conditions at cells which coincides with or touch the boundary domain.

The solution to a flow problem (velocity, pressure, temperature etc.) is defined at nodes inside each cell. The accuracy of a CFD solution is governed by the number of the cells in the grid. Larger the number of cells, better the solution accuracy.

2. Solver

The solver basically performs the following steps:

1. Approximation of the unknown flow variables by means of simple functions.
2. Discretization by substitution of the approximation into the governing flow equations and subsequent mathematical manipulations.
3. Solution of the Algebraic equations.

3. Post Processor

This is the tool used for examining the results such as

1. Vector plots
2. 2D & 3D surface plots
3. Contours plots with different colors.

6.4 Governing Equation:

The following governing equations were solved for each discretized element:

Continuity equation:

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}=0 \tag{6.1}$$

X-Momentum equation:

$$u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}+w\frac{\partial u}{\partial z}= -\frac{1}{\rho}\frac{\partial P}{\partial x}+v\left[\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}+\frac{\partial^2 u}{\partial z^2}\right] \tag{6.2}$$

Y-Momentum equation:

$$u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}+w\frac{\partial v}{\partial z}= -\frac{1}{\rho}\frac{\partial P}{\partial y}+v\left[\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}+\frac{\partial^2 v}{\partial z^2}\right] \tag{6.3}$$

Z-Momentum equation:

$$u\frac{\partial w}{\partial x}+v\frac{\partial w}{\partial y}+w\frac{\partial w}{\partial z}= -\frac{1}{\rho}\frac{\partial P}{\partial z}+v\left[\frac{\partial^2 w}{\partial x^2}+\frac{\partial^2 w}{\partial y^2}+\frac{\partial^2 w}{\partial z^2}\right] \tag{6.4}$$

Energy Equation:

$$\left[u\frac{\partial T}{\partial x}+v\frac{\partial T}{\partial y}+w\frac{\partial T}{\partial z}\right]=\alpha\left[\frac{\partial^2 T}{\partial x^2}+\frac{\partial^2 T}{\partial y^2}+\frac{\partial^2 T}{\partial z^2}\right] \tag{6.5}$$

6.5 COMPUTATIONAL DOMAIN

In this study, computational process is carried out using FLUENT, a commercial CFD code which is based on finite volume method. CFD is used to predict internal and external flow which results in valuable time reduction and new design cost as well,

Following assumptions are made in present simulations: (1) Flow and heat transfer is steady and turbulent (2) Working fluid is incompressible (3) Tube wall temp kept constant in the whole shell side. (4)The heat exchanger is well insulated; hence the heat loss to the environment is totally neglected.

6.5.1 Periodic Boundaries

According to periodic geometries of STHXs, it is convenient to study one period of whole heat exchanger. The periodic model is shown in "Figure 6.3".In the present study, one period of complete heat exchanger have been simulated. According to model in a periodic form, baffle pitch and boundaries are the same in size and locations.

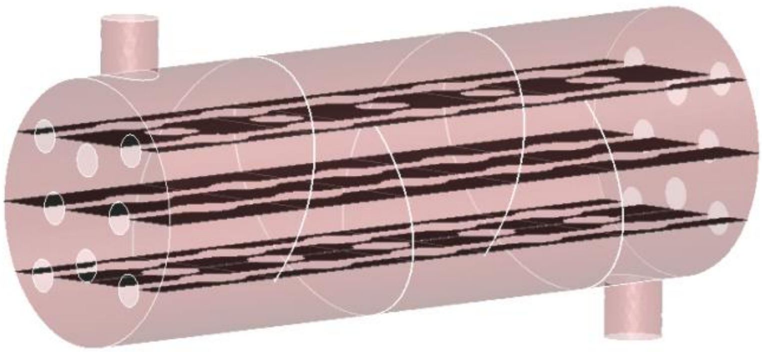

Figure 6.3 Periodic model of STHXs

6.5.2 Mesh Generation

The computational domains are meshed with unstructured Tri-tet grid which is suitable for simulating complex geometry. For mesh generation, one period of whole heat exchanger is taken because the results for each period are approximately same.

Surface mesh is created with the triangular elements and "Patch dependent" as mesh algorithm. Here the main objective to model the shell side flow and volume mesh is done only inside the shell using tetrahedral elements & Delaunay as algorithm. Total trias mess element is 46282 and tetras are 236388 observed from mesh info. Gradients are very high near the wall due to thermal and boundary layers. To include the effect of those boundary layer prism layers are taken on the wall boundaries

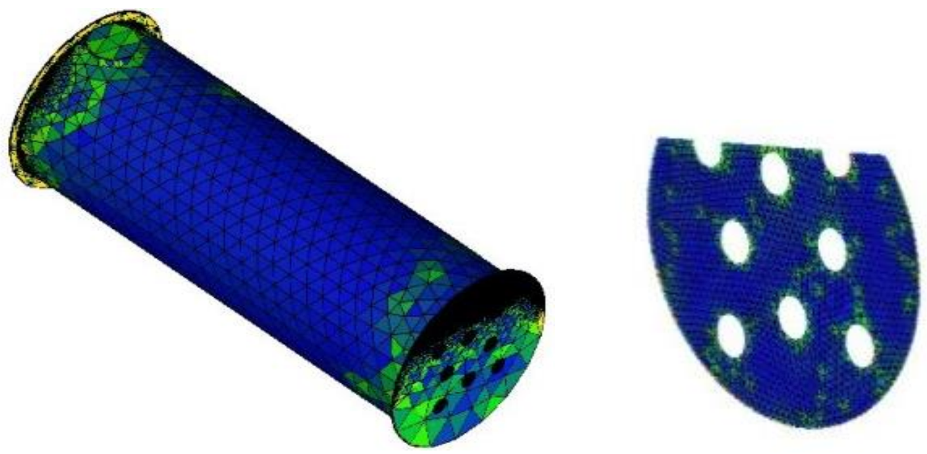

Figure 6.4 Surface mesh on lumped tubes and baffles

.

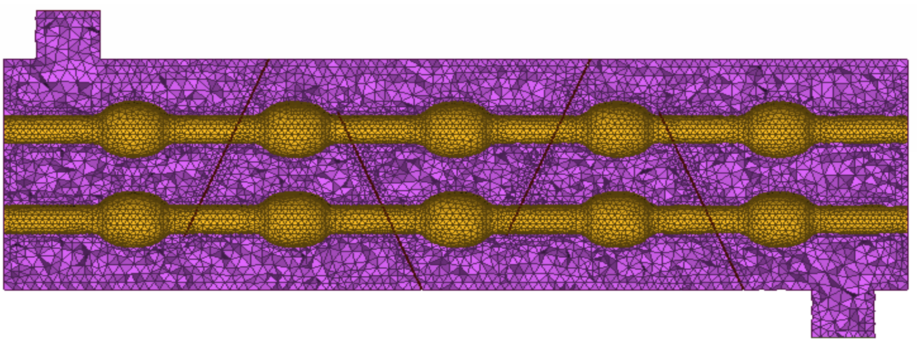

Figure 6.5 Volume Meshed on Periodic Model

6.5.3 Prism Layers:

Gradients are very high near the wall due to thermal and boundary layers. To include the effect of those boundary layer prism layers are taken on the wall boundaries.In this case three surfaces are treated as a wall boundaries in that shell wall, all tubes OD & baffles.

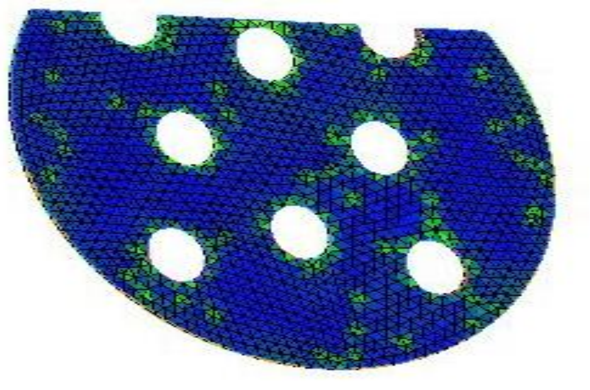

Figure 6.6 Tube layout pattern with prism layers

6.5.4 Turbulence model

In order to simulate the flow and heat transfer a three dimensional ķ-ε realizable turbulence model is applied. Compared with the ķ-ε turbulence models, an immediate benefit of the ķ-ε realizable turbulence model is that it makes more accurate predictions. Near wall treatment is selected as an enhanced wall treatment because Y-plus value <1. Y –plus gives the boundary effect near the wall. Y+ values play vital role in turbulence modeling for the near wall treatment.

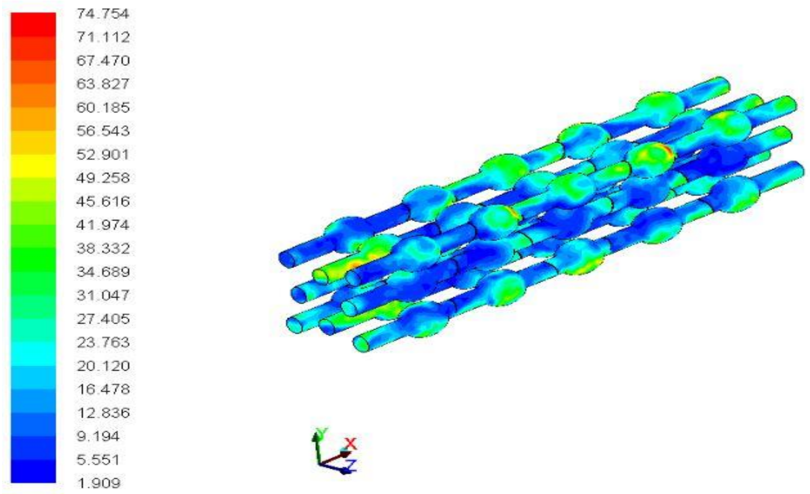

Figure 6.7 Y^+ on tube bundles and baffle walls

6.5.5 Boundary conditions

Boundary conditions used according to the requirement of model. The tubes and shell walls are separately defined with respective boundary conditions. In the periodic models, 320# conductive oil taken as a working fluid. Shell wall considered as stationary wall and applied no slip boundary conditions. Shell wall is adiabatic i.e.; wall heat flux 0.

Table 6.2 Boundary conditions.

Boundary	Boundary condition type	Parameters
Inlet shell	Mass flow inlet	3.54, 5, 8, 10, 12 Kg/s & 70°C
Outlet shell	Pressure outlet	0 Pa (Gauge pressure)
Shell Walls	No slip conditions	0 Wall-Heat flux
Tube walls	Wall-constant temperature	40°C

In helical baffle thin- walls method are applied and made of steel and its thermal conductivity is taken as 15.2 W/m-K.

6.5.6 Grid Independency Test

In order to obtain more accurate results for heat transfer coefficient grid independency test is conducted. Three different grid systems are selected for the model DCH-STHXs to obtain accurate results. “Figure 6.8” shows the variation in heat transfer coefficient with different cell number 275000, 282000, 370000 are taken and it is observed with increasing cell numbers heat transfer coefficient increase. The percentage deviation in heat transfer coefficient is 2% for the last two grid cells. Thus second grid system is selected for computational domain to analysis.

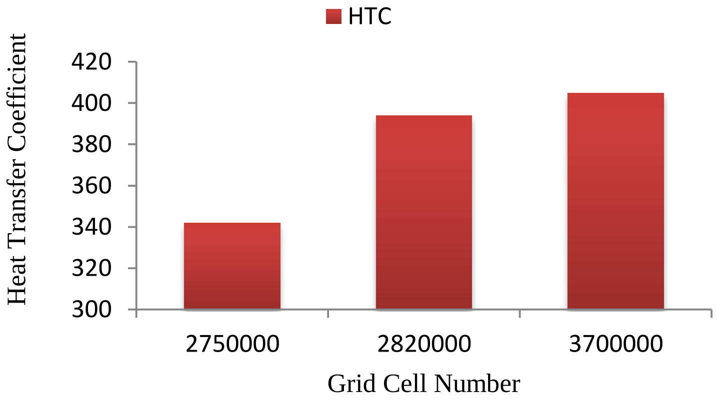

Figure 6.8 Grid independency test

6.6 Method of Simulation

The computational process is carried out using FLUENT, a commercial CFD code which is based on the finite volume method. The governing equations are iteratively solved by using SIMPLE pressure- velocity coupled algorithm. This numerical approach stores scalar variables at the center of the control volume. The second order upwind scheme is used for the numerical simulation.

6.6.1 Convergence criteria

As for the convergence criterion, the sum of the normalized absolute residuals in each control volume for energy variable (T) is < 10^{-6}, because flow variable are much more difficult to converge. Each solution takes approximately 3-4 hours to converge on personal computer having 4 GB RAM & i-3 core processor.

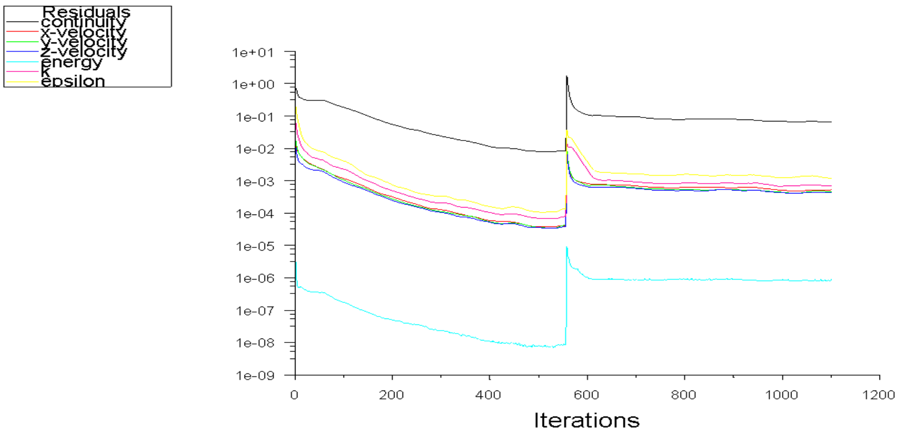

Figure 6.9 Residual convergence

6.6.2 Solution procedure overview

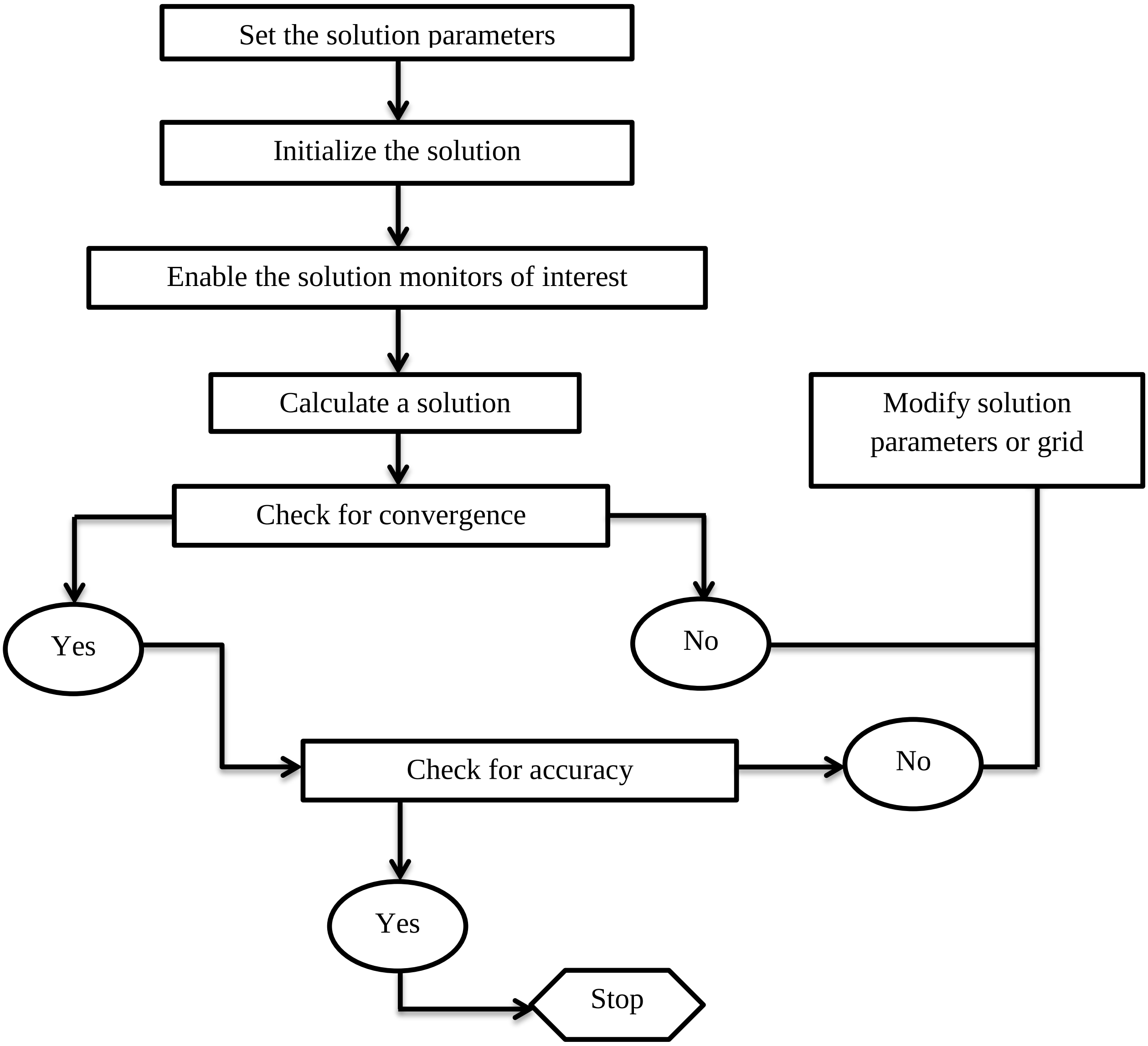

Figure 6.10 Flow chart of solution procedure

6.7 Data Reduction

Main parameters and equations are used to calculate fluid flow and heat transfer performance

Shell side heat transfer rate

$$Q = M.C_p.\left(T_{s,in} - T_{s,out}\right) \tag{6.6}$$

Shell side heat transfer coefficient

$$h = \frac{Q}{A_0.\Delta T_m} \tag{6.7}$$

Heat transfer surface area

$$A_0 = N.\pi.d_0.L \tag{6.8}$$

Logarithmic mean temperature difference

$$\Delta T_m = \frac{\Delta T_{max} - \Delta T_{min}}{\ln(\frac{\Delta T_{max}}{\Delta T_{min}})} \tag{6.9}$$

Where,

$\Delta T_{max} = T_{s,in} - T_w$

$\Delta T_{min} = T_{s,out} - T_w$

Total heat transfer rate, pressure drop, outlet bulk temperature are provided through CFD software.Where T_w is temperature of the wall of the tube and A_0 is the heat transfer area based on outside diameter of tube bundles.

6.8 Numerical Validation

Heat transfer coefficient and pressure drop are our main concern for designing heat exchanger. In traditional Bell-Delaware method is used to validate the results which are obtained from commercial code fluent analysis. Thermal analysis of shell and lumped tube heat exchanger with helical baffles hasn't been done using Bell-Delaware method. Bell-Delaware method is proven & has been verified by other researchers

To validate heat transfer coefficient results

$$h_0 = h_{ideal}(J_c J_l\ J_{b)} \tag{6.10}$$

Shell side pressure drop

$$\Delta P_s = [(n_b - 1) \times \Delta P_c \times R_B + n_b.\Delta P_w]R_L + 2\Delta P_c\left(1 + {}^{N_{cw}}/_{N_c}\right)R_B \tag{6.11}$$

Cross flow area

$$s_m = D[(D_s - D_{out}) + \frac{(D_{out} - D_0)}{P_t} \tag{6.12}$$

"Figure 6.11" and "Figure 6.12" shows the validation curve, in order to validate the simulated results. The maximum differences between present simulation and numerically calculation are around and 5.5% respectively for the pressure drop and 7.2% and for the heat transfer coefficient. It is observed that results of present analysis came in close agreement with the aforementioned researches hence current model is efficient enough to be analyzed. Bell-Delaware method is used for calculation of predicted values, which is used to compare this values with simulated values which drawn from simulation. The graph below shows the variation of heat transfer coefficient and pressure drop as a function of mass flow rate.

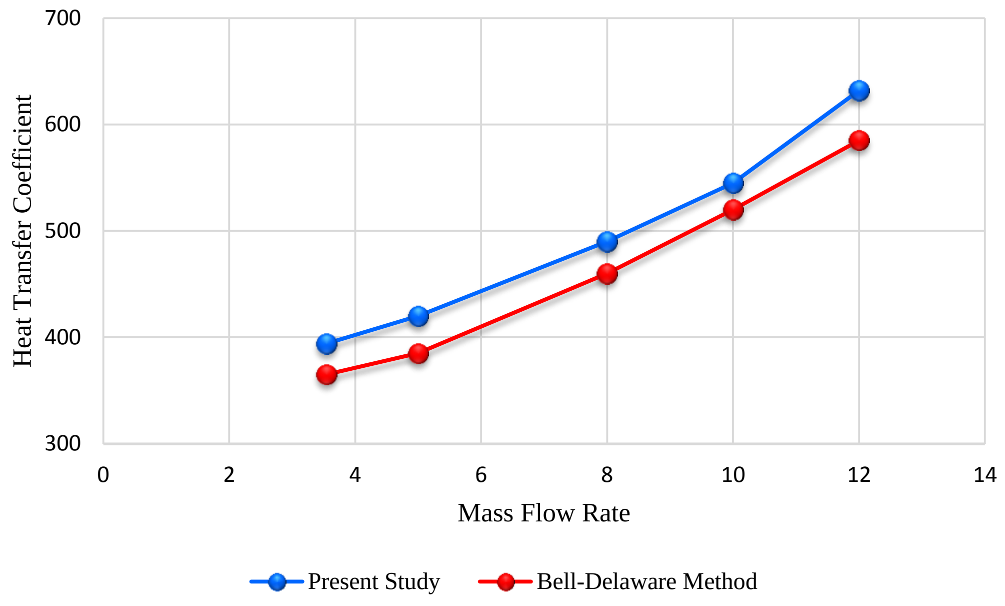

Figure 6.11 Numerical Validation of results HTC

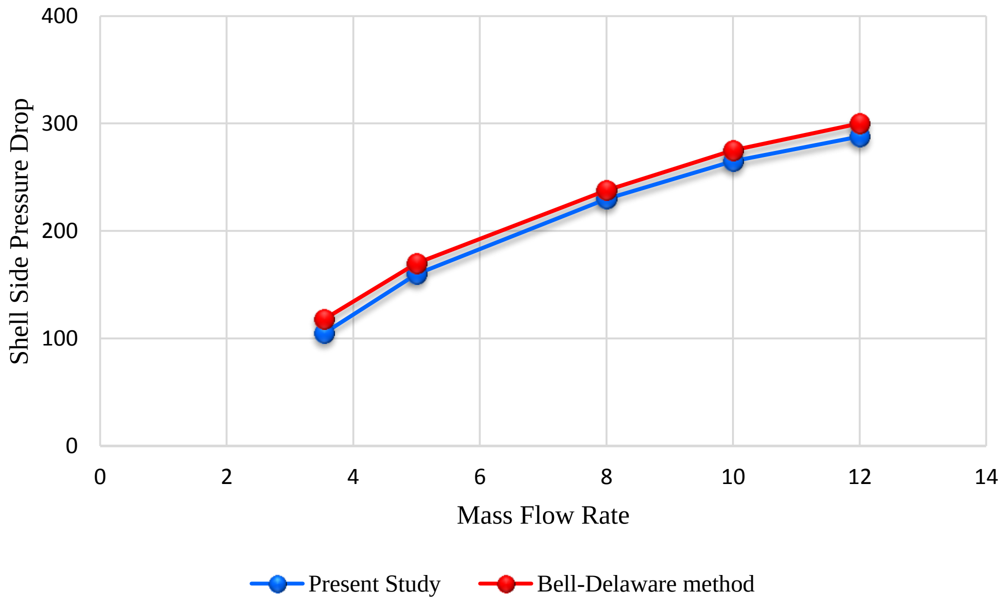

Figure 6.12 Numerical Validation of results ΔP.

7.1 Pressure Drop and HTC

Pressure drop played an importance role in design of STHXs because pumping power is directly depend upon pumping cost results in lower pressure drop leads to lower operating cost. "Figure 7.8" indicates the shell side pressure drop vs. mass flow rate and it is observed that better performance with compared to conventional STHXs. It is observed that in the shell side, the fluid passes through the tube bundle in a zigzag manner which results in high heat transfer coefficient due to turbulences in the flow. For the discontinuous helical baffle fluid flows in a helical pattern that cause's lower pressure drop. So there is a good result shown by two character the heat transfer coefficient and pressure drop.

The result shows that the heat transfer coefficient increases with decrease of the helix angle. This variation trend can be understood from two aspects 1) At fixed shell inner diameter, the helix pitch and cross flow area decreases with decrease of helix angle and at same mass flow rate the shell side velocity increases as with the decrease of cross flow area. Thus the convective heat transfer is enhanced with the decrease of the helical angle because of increased velocity. 2) With the decrease of helical angle, the flow pattern of shell side fluid flow gradually approaches the external flow cross flow to the tube bank, which results in better heat transfer intensity than the flow parallel to tube bank at the same velocity.

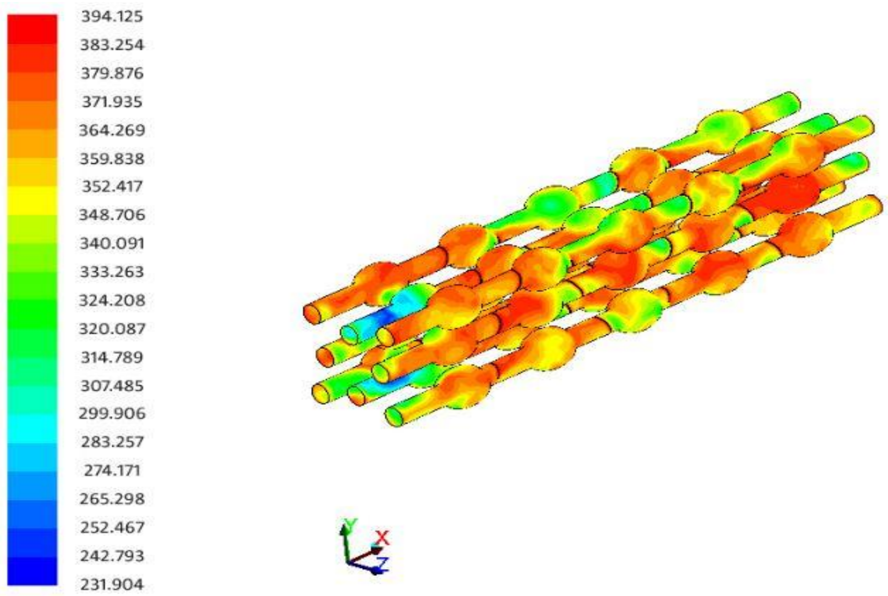

Figure 7.1 Heat Transfer Performance on Lumped Tubes Bundle.

From the earlier research it is found that 40° is the optimum helix angle where it gives high heat transfer coefficient per unit pressure drop. If we talk about only heat transfer coefficient it is found that it decrease with increase in helix angle i.e. at lower helix angle it gives good heat transfer coefficient value. This is understood by decrease in helix angle at the same mass flow rate the shell side velocity increases as with the decrease of flow area. “Figure 7.1” shows the convective heat transfer is increased with the 22° helix angle which gives better heat transfer performance per unit pressure drop because of increased velocity. So the STHXs with lumped tube and helical baffles give better performance than DCH-STHXs and conventional STHXs.

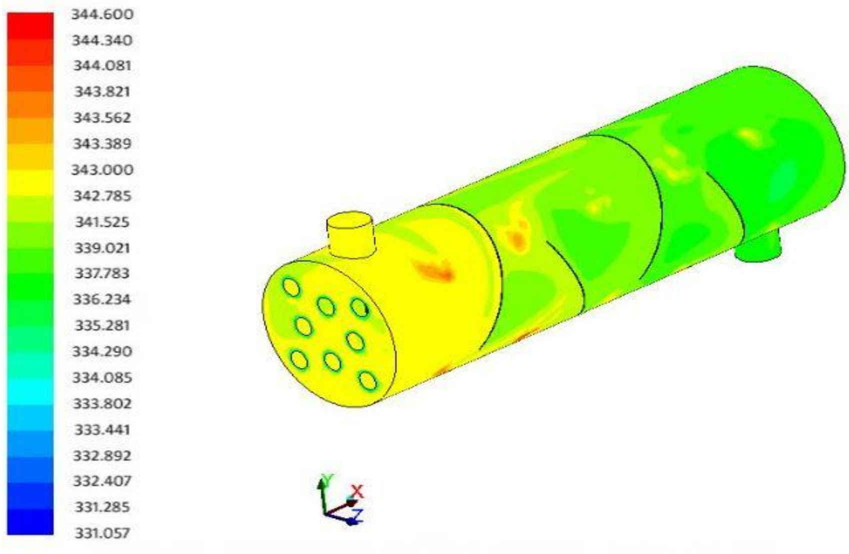

Figure 7.2 Temperature Variation in Shell Side

“Figure 7.2” shows the temperature of shell side flow decrease as it flows over the tubes having constant temperature boundary conditions. As shell side water flows over tubes it gets cooled to release the heat with the tube walls & shell side water temperature decrease. Because of turbulence created in shell side flow heat transfer is enhanced in shell region. As contour shows the effect of temperature variation spread inside & along the periphery of the shell.

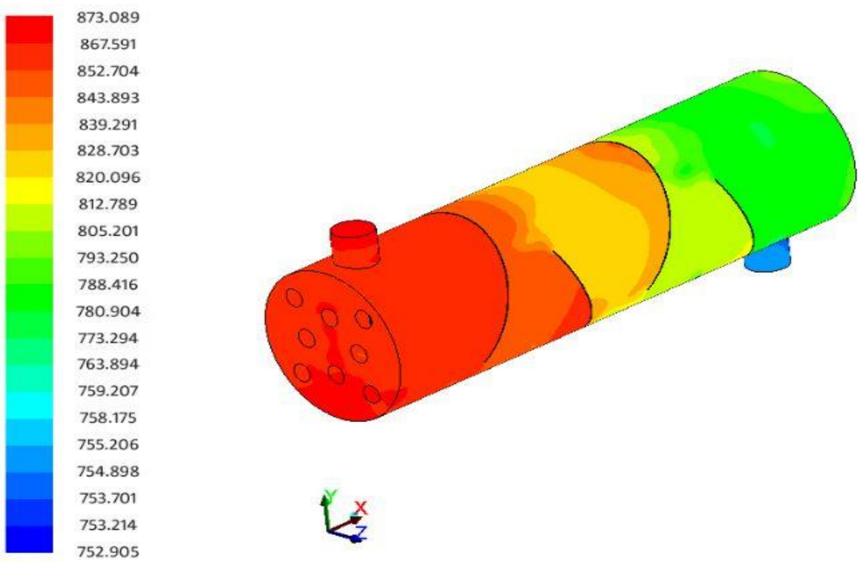

Figure 7.3 Pressure Variation in Shell Side

"Figure 7.3" shows that, when the shell side fluid flows at inlet, it shows the higher value of pressure and as it flows over the tubes in a helical path and zigzag path at the outlet, it shows the lower pressure as the flow passes from inlet to outlet, the region between inlet and outlet shows pressure drop because of the flow striking the baffles and thus reducing the velocity.

"Figure 7.3" shows the pressure drop contour in STHXs for full model. As flow passes through each helix cycle and zigzag manner it shows low pressure drop. Finally the fluid leaves at outlet shows pressure drop for 3.54 kg/sec mass flow and 22° helix is approximately 120Pa.

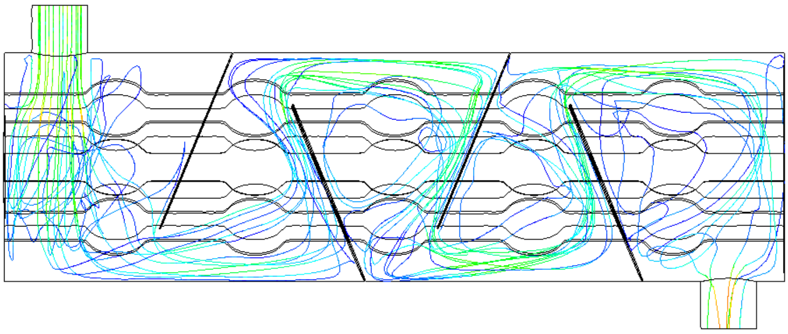

Figure 7.4 Velocity Streamlines on STHXs

Shell side flow vector and streamlines are shown in "Figure 7.4". The shell side flow pass basically in good pattern and rushes along with the helical baffle. Due to lumped tube flow avoids abrupt turns of flow. Therefore it enhances the heat transfer coefficient per unit pressure drop.

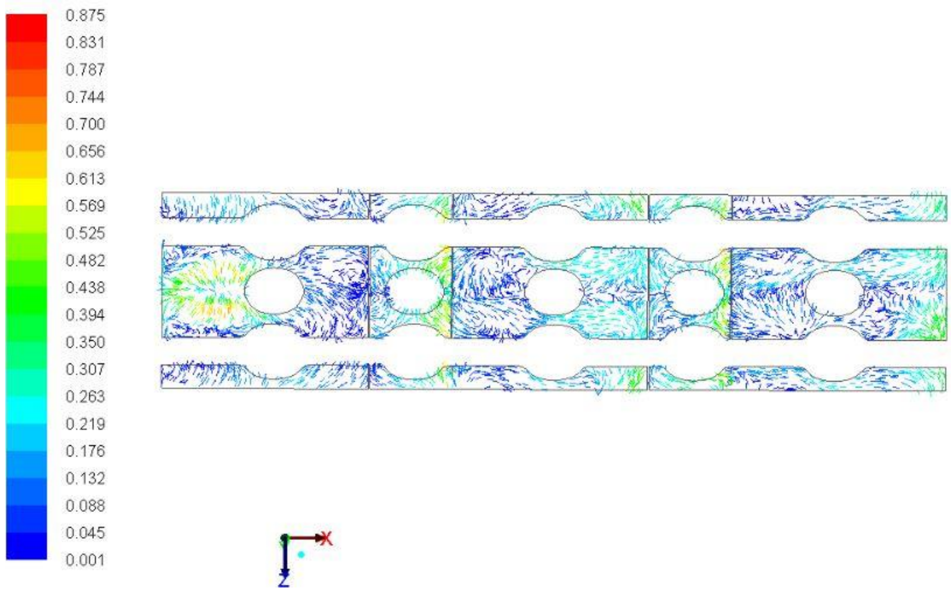

Figure 7.5 Vector plot in Shell and Lumped Tube Heat Exchanger

Above “Figure 7.5” shows the vector plot of shell and lumped tube heat exchanger in Z-X plane. It is shows that the fluids follow the helical path. Baffles are meant to guide the shell side flow and support the tubes. It is also shows the behavior of the fluid in shell side.

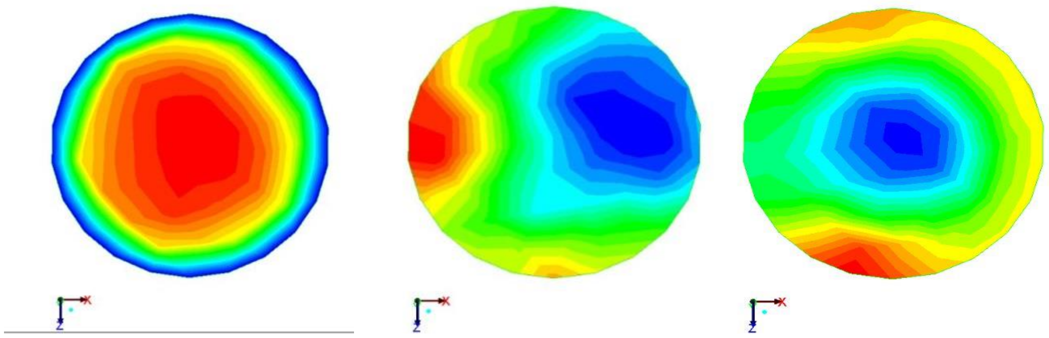

Figure 7.6 Outlet Velocity and Outlet Pressure and outlet Temperature variation in mid cross section

Outlet velocity, outlet pressure and outlet static temperature distribution are shown in “Figure 7.6” above, which is helps to predict the flow inside and outside the lumped tubes. This provides the clear idea about velocity, pressure and static temperature at outlet of the shell and tube heat exchanger.

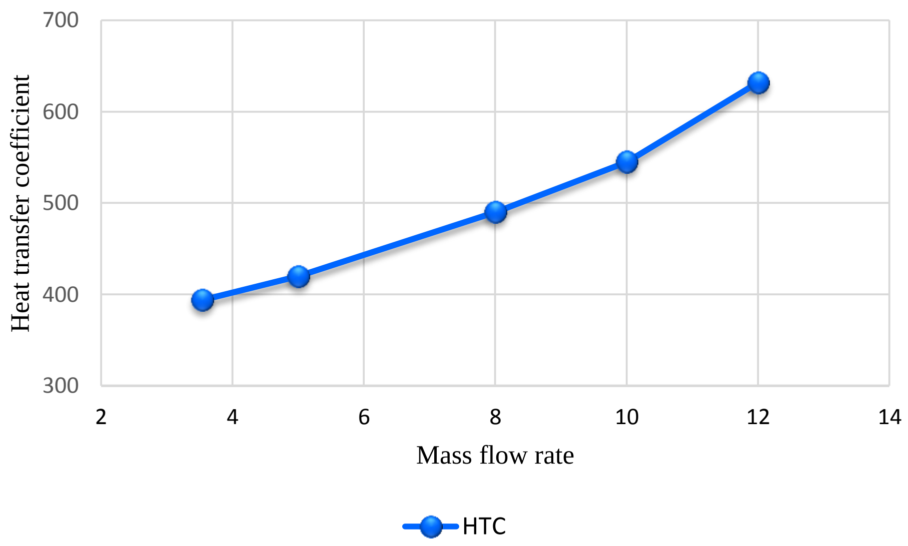

Figure 7.7 Heat transfer coefficient v/s Mass flow rate

“Figure 7.7” shows the variation in shell side heat transfer coefficient with the mass flow rate and it is observed that as the mass flow rate increases heat transfer coefficient is also increases due to increase in shell side velocity.

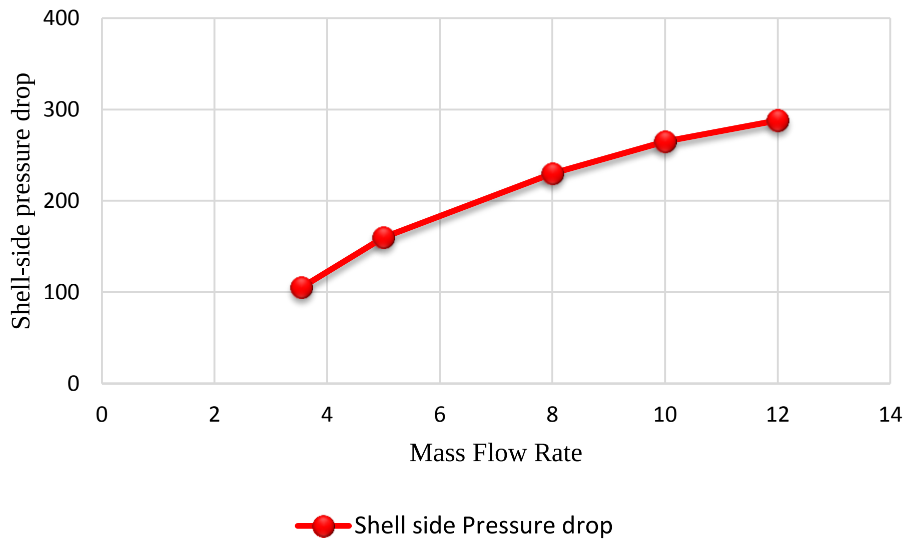

Figure 7.8 Shell side Pressure drop v/s Mass

"Figure7.8" shows the variation in shell side pressure drop with the mass flow rate and it is observed that as the mass flow rate increases shell side pressure drop ΔP also increases.

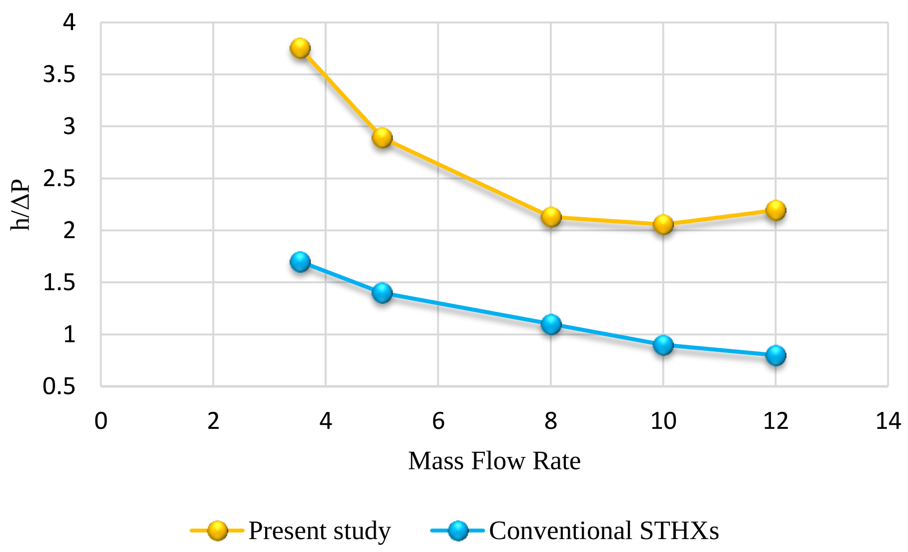

Figure 7.9 h/ΔP v/s Mass flow rate

It is observed that from the previous study that as we increase helix angle heat transfer is decrease this is because as we increase helix angle cross flow area increases and decrement in shell velocity that reduces the heat transfer coefficient but it is also observed that as we increase in helix angle pressure drop is reduces. So I established a proper combination between heat transfer coefficient and pressure drop in which it gives its optimum value. So a figure shows the variation in h/ΔP with the mass flow rate and it is found that it gives better performance than the conventional shell and tube heat exchanger.

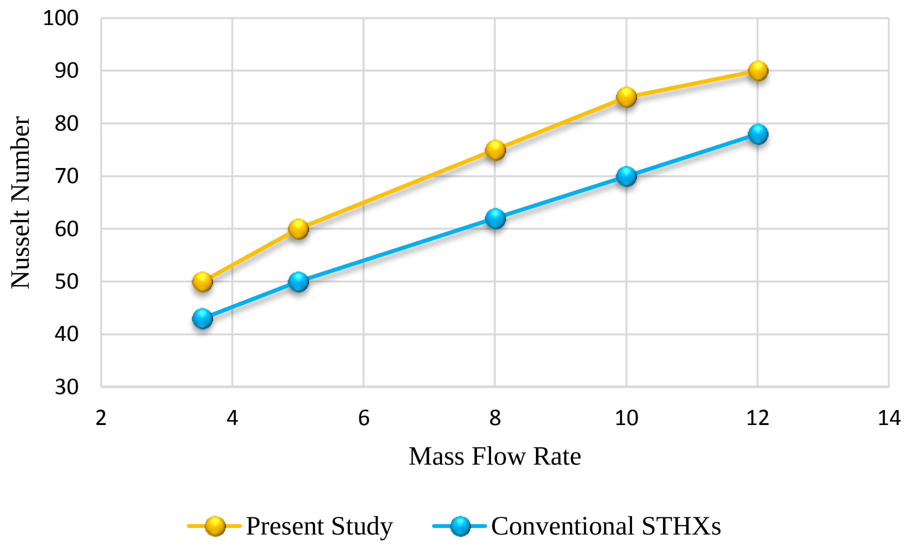

Figure 7.10 Nusselt number v/s Mass flow rate

“Figure 7.10” shows the variation in shell side Nusselt number versus Mass flow rate M, the Nusselt number 9.3-30.2% higher than those conventional STHXs. This is because shell side flow is diverted to the tube bundles which increase the contact time of fluids with each other and it leads to increase the heat transfer coefficient.

8.1 Conclusion

In this project computational simulations of shell and lumped tube heat exchanger with helical baffles are performed to reveal the effect of lumped tube on the heat transfer and pressure drop characteristics. It is found that its increase its heat transfer per unit pressure drop due to increase in contact surface area and contact time between the fluids. This type of geometry also lead to increase the turbulence of the flowing fluids and it will generate secondary flow inside the tube and better mixing causes increases its heat transfer efficiency for the same volume of heat exchanger.

Present work reported the effect of lumped tube and helical baffle in STHXs. In this study 3D numerical simulations are carried out in shell and lumped tube heat exchanger. It is found that with the lumped tube and helical baffle heat transfer coefficient per unit pressure drop increases at higher mass flow rate. The main results are summarized follows:

(1) Under the same mass flow rate Heat transfer coefficient per unit pressure drop is increased 30-33% as compared to conventional shell and tube heat exchanger (STHXs).

(2) At same mass flow rate and same working condition, present work shows greater heat transfer coefficient and lower pressure drop over segmental baffle shell and tube heat exchanger.

(3) Under the same mass flow rate Nusselt number is 9.3-30.2% higher than that conventional STHXs and the friction factor is also increase but in very less amount.

8.2 Future Scope

1) There is a possibilities to adopt new passive techniques in shell and tube heat exchanger and compared with present model.
2) It will be used in industries for company’s growth with the minimum operating cost.
3) It can be done by mathematical modeling and experimental work.
4) There is scope to create roughness over the entire tube which gives better turbulences in the fluid and gives better performance.

REFERENCES

[1] C.J. Bolinder, “Curvilinear coordinates and physical components: an application to the problem viscous flow and heat transfer in smoothly curved ducts”, J. Appl. Mech. (63) (1996) pp.985–989.

[2] Chen Gui-Dong, Ming Zeng, and Qui-wang. “Experimental and Numerical studies on shell side performance of three different shell and tube heat exchangers with helical baffles”, Journal of Enhanced Heat Transfer,(2011) pp.449-463,.

[3] Ender Ozden. “Shell side CFD analysis of a small shell-and-tube heat exchanger”, Energy Conversion and Management 51 (2010) 1004–1014.

[4] Farhad Nemati Taher, Sirous Zeyninejad Movas, “Baffle space impact on the performance of helical baffle shell and tube heat exchangers”, Applied Thermal Engineering 44 (2012) 143-149.

[5] Gopi Chand, “simplified model for the study of thermal analysis of shell-and-tubes heat exchangers of water and oil type”, IEEE (2011) pp.855-863

[6] Gh.Sumit. Jahanmir, F. Farhad, “investigated twisted bundle heat exchangers performance evaluation by CFD”, International Communications in Heat and Mass Transfer 39 (2012) 1654–1660.

[8] Jian-Feng Yang, Min Zeng, Qiu-Wang Wang, “Effects of sealing strips on shell side flow and heat transfer performance of a heat exchanger with helical baffles”, Science direct 64 (2014), pp.117-128.

[9] Jie Yang, Lei Ma have been studied a “comparison of four numerical modeling approaches for enhanced shell-and-tube heat exchangers with experimental validation”, Applied Thermal Engineering 65 (2014), pp.369-383

[10] Jian-Fei Zhang, “Experimental performance comparison of shell-side heat transfer for shell-and-tube heat exchangers with middle-overlapped helical baffles and segmental baffles”, Chemical Engineering Science 64 (2009) 1643--1653

[11] Lin Guang-yi, Zhang Xiao-hong, Liu Ze Yang Wei-min, “Research on enhanced heat transfer characteristics in heat exchanger inserted with rotors”, IEEE (2011)

[12] Mohnish Salimpour, “Heat transfer coefficients of shell and coiled tube heat exchangers”, Experimental Thermal and Fluid Science 33 (2009), pp. 203–207.

[13] Mao-Yu Wen studied “Heat-transfer enhancement in fin-and-tube heat exchanger with improved fin design”, Applied Thermal Engineering 29 (2009) 1050–1057.

[14] Mohammadi Koorosh, “Investigation of Baffle Cut and Fluid Viscosity”, Abadam; 2011

[15] Mr.Pravin kumar, V. Hadgekar. "Comparative thermal analysis of conventional tublar heat exchanger using Bell-delaware method." International Journal of Scientific and Research Publications; 2013.

[17] Qiuwang Wang “Numerical investigation on combined multiple shell-pass shell-and-tube heat exchanger with continuous helical baffles”, International Journal of Heat and Mass Transfer 52 (2009) 1214–1222.

[18] Shinde S. Sunil studied “Numerical comparison on Shell side performance of Helixchanger with center tube with different helix angles”, International Journal of Scientific and Research Publications, Volume 3, Issue 8, August 2013 ISSN 2250-3153.

[19] Shinde S. Sunil has studied about “the performance Improvement in Single phase Tubular Heat Exchanger using continuous Helical Baffles”, IEEE (2012).

[20] Simin Wang, “experimental investigation of heat transfer enhancement for a shell-and-tube heat exchanger”, Applied Thermal Engineering 29 (2009) pp. 2433–2438.

[22] Tan Wee Choon, Mohd Shahril Shariff, Matthew Law, “Computational Fluid Dynamics Analysis of Shell-and-double Concentric-tube Heat Exchanger”, IEEE (2011).

[23] Xiang-hui Tan, “3D numerical simulation on the shell side heat transfer and pressure drop performances of twisted oval tube heat exchanger”, International Journal of Heat and Mass Transfer 65 (2013),pp. 244–253.