ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

Numerical Analysis of Shell and Tube Heat Exchanger with Different Proportions of Glycerine

G. Sesi Bhusana Rao
Department of Mechanical Engineering,
Sri Vaishnavi College of Engineering,
Singupuram, A.P - 532 185, India.

S. Suresh Babu
Department of Mechanical Engineering,
Sri Vaishnavi College of Engineering,
Singupuram, A.P - 532 185, India.

ABSTRACT

In present days shell and tube heat exchanger is the most common type heat exchanger, it is widely used in oil refinery and other large chemical processes, because it is suitable for high pressure applications.

The process in solving simulation consists of modelling and meshing the basic geometry of shell and tube heat exchanger using CFD package ANSYS 15.0. Shell and Tube Heat Exchanger (STHE) has its own importance in the industrial applications .The improvement of efficiency of the STHE using different design modifications in the shell side and different concentrations of Nano fluids results in increase of overall efficiency. In that view, Experimental and numerical simulations are carried for a single shell and multiple pass heat exchanger with different glycerin percentages. The experiment was carried out with hot fluid as water in tube side and cold fluid as water in shell side with circular tubes at 600 tube orientation and 25% baffle cut. Heat transfer rates and temperatures are calculated for various Reynolds numbers from 4000 to 20000. Fluent software is used for numerical investigations. Both water and different concentrations i.e. 20%, 40%, 60%, 80% and finally 100% of glycerin are used for the numerical studies. In addition to 25% baffle cuts are used for comparison.

The experimental values of heat transfer rates and velocities over shell side and tube side along the length of STHE are compared with those obtained from fluent software. It is found that the glycerin with 20% concentration will give maximum heat transfer rates and lower pressure drops.

Key words: *heat exchangers, cfd, catia, helical, inclined baffles*

1. INTRODUCTION

Heat Exchanger

Heat exchanger is a universal device in many industrial applications and energy conversion systems. Various heat exchangers are designed for different industrial processes and applications. In heat exchangers, shell and tube heat exchanger presents great sustainability to meet requirements and gives efficient thermal performance.

STHE are widely used in petro-chemical industry, Power generation, and energy conservation and manufacturing industry [1]. The baffle member plays an important role in STHE and it supports tube bundle and also equally distribute the fluid in shell side. When segmental baffles are used in STHE which have many disadvantages .The low heat transfer is achieved due to the flow stagnation i.e., dead zones which are created at the corners between baffle and shell wall. It requires higher pumping power and it creates high pressure drop under the same heat load. The orientation of tubes will influence the annular surface area surrounded by the fluid. It is also influences the heat transfer rate. New baffle cut arrangement is achieved higher heat transfer rates and lower pressure drops , so it is required to develop a new type STHE using different baffle cut arrangement to achieve higher heat transfer rate. The last few years which describe methods to calculate heat transfer and pressure drop in

Cite this article as: G. Sesi Bhusana Rao & S. Suresh Babu, "Numerical Analysis of Shell and Tube Heat Exchanger with Different Proportions of Glycerine", International Journal & Magazine of Engineering, Technology, Management and Research, Volume 5 Issue 2, 2018, Page 118-130.

shell side of STHE with different baffles. The different calculation procedures have been checked in against an experimental measurements on small scale heat exchanger. The Kern method, Tinker method and Delaware method gave the best results. Compared with other methods in literature. The shell side design under the flow phenomenon inside the shell must be understood by experimental and numerical analysis. The shell and tube heat exchanger design explained by Gay et al worked on heat transfer, while Halle et al Pekdemir et al investigated pressure drop. Now a days the numerical methods have become an economical alternative for the research of STHE, through which detailed flow pattern and temperature field could be obtained with much less difficult. In order to attain better performance characteristics (high heat transfer rates and low pressure drops) the STHE have to maintain the geometry of the heat exchanger and heat transfer fluids (Nano fluids Fluid which consisting of nano size particles is a Nano fluid and the fluid is dispersed in a conventional fluid. Basically used conventional fluids are water, engine oil, ethylene glycol have low thermal conductivity relative to metal and even metal oxides [2].

Use of Baffles:

The main roles of a baffle in a shell and tube heat exchanger are to: Hold tubes in position (preventing sagging), both in production and operation prevent the effects of vibration, which is increased with both fluid velocity and the length of the exchanger Direct shell-side fluid flow along tube field. This increases fluid velocity and the effective heat transfer co-efficient of the exchanger in a static mixer, baffles are used to promote mixing.In a chemical reactor, baffles are often attached to the interior walls to promote mixing and thus increase heat transfer and possibly chemical reaction rates [3].

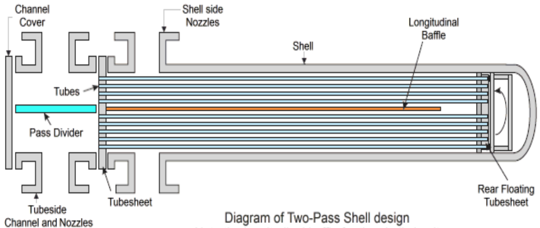


Diagram of Two-Pass Shell design
Note the longitudinal baffle for the shell circuit

Fig 1.1: Double pipe heat exchanger

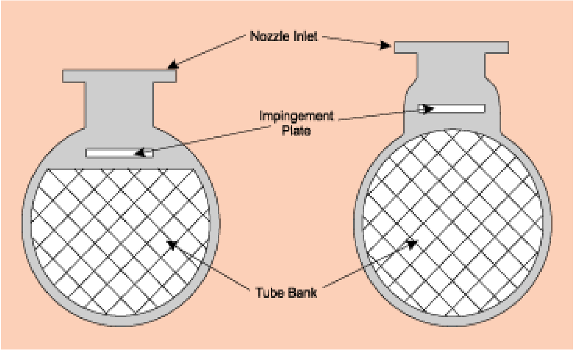


Fig 1.2: Impingement Baffles

1.3 Heat Transfer Considerations

The energy flow between hot and cold streams, with hot stream in the bigger diameter tube, is as shown in Figure 2.10. Heat transfer mode is by convection on the inside as well as outside of the inner tube and by conduction across the tube. Since the heat transfer occurs across the smaller tube, it is this internal surface which controls the heat transfer process [4]. By convention, it is the outer surface, termed Ao, of this central tube which is referred to in describing heat exchanger area. Applying the principles of thermal resistance,

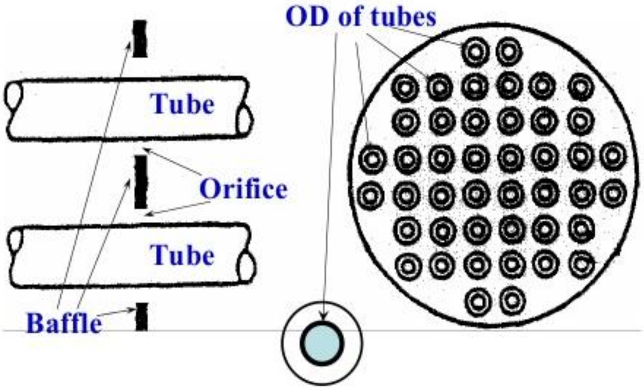


Fig 1.3: Orifice Baffles

$$R = \frac{1}{h_o A_o} + \frac{\ln\left({r_o}/{r_i} \right)}{2\pi \cdot kl} + \frac{1}{h_i A_i}$$

If we define overall the heat transfer coefficient, Uc, as:

$$U_c = \frac{1}{RA_o}$$

Substituting the value of the thermal resistance R yields:

$$\frac{1}{U_c} = \frac{1}{h_o} + \frac{r_o \ln\left({r_o}/{r_i} \right)}{k} + \frac{A_o}{h_i A_i}$$

Standard convective correlations are available in text books and handbooks for the convective coefficients, *ho* and *hi*. The thermal conductivity, *k*, corresponds to that for the material of the internal tube. To evaluate the thermal resistances, geometrical quantities (areas and radii) are determined from the internal tube dimensions available [5].

1.4 Log Mean Temperature Differences

Heat flows between the hot and cold streams due to the temperature difference across the tube acting as a driving force. As seen in the Figure 7.3, the temperature difference will vary along the length of the HX, and this must be taken into account in the analysis.

1.5: Temperature Differences between Hot and Cold Process Streams

From the heat exchanger equations shown earlier, it can be shown that the integrated average temperature difference for either parallel or counter flow may be written as:

$$\Delta\theta = LMTD = \frac{\theta_1 - \theta_2}{\ln\left(\dfrac{\theta_1}{\theta_2} \right)}$$

The effective temperature difference calculated from this equation is known as the log mean temperature difference, frequently abbreviated as LMTD, based on the type of mathematical average that it describes. While the equation applies to either parallel or counter flow, it can be shown that Δθ will always be greater in the counter flow arrangement. Another interesting observation from the above Figure is that counter flow is more appropriate for maximum energy recovery [6].

2.LITERATURE REVIEW

Joydeep Barman et.al. [1] Designed the optimum shell and tube heat exchanger where, shell diameter is constant and longitudinally finned tube with different pitch values. They mentioned that by using analytical calculations with kern method and to determine the behaviour on varying of the heat exchanger with their variations for both of square and triangular pitch arrangements, along with values of pressure drop.

The maximum heat transfer rate was calculated for optimum pin height and same diameter of shell and different pitch arrangements were considered by them. They stated that to increase the number of tube side passes at constant shell diameter heat transfer rate increases, pressure drop decreases. They stated that, if the number of tube passes decreases the performance heat exchanger decreases and increased pressure drop.

M.R. Salimpour et.al [2] investigated the heat transfer coefficients of shell and coil type heat exchanger. They analyzed thermal performance of three different helical coil pitches of heat exchanger, conducted by parallel and counter flow arrangement. They said that the coils with larger pitches gives better heat transfer coefficients than smaller coil pitch value. In this coil tube heat exchanger inner and outer tube heat transfer coefficients are predicted by using two correlations. The anticipated qualities are in great concurrence with exploratory results.

Jian-Fei Zhang et al. [3] Compared a shell and tube heat exchangers with segmental and helical baffles at different helix angles of 200, 30 0,400 and 500, respectively. They detailed that the heat transfer coefficient increase with helical baffle as compare to segmental baffle of shell and tube heat exchanger. The heat duty is calculated at same shell in side diameter and at altered helix angles by them. The heat transfer rate upsurges with rise of helix angle. They stated that the helical baffle with 50◦ helix angle shows the better heat transfer rate than other helix angles.

F.Vera-Garcia et al [4] carried out experimental analysis to calculate the heat transfer coefficient of shell and tube heat exchanger is working as condenser and evaporator. They said that this whole experimental arrangement acts as a complex refrigeration system. They used R22 as refrigerant and conducted number of experiments in a three levels of condensation pressure and three levels of evaporation pressure. The focal objective of the model is to provide the outlet conditions, where the inlet conditions are known as part of the modelling procedure. The experimental and computational results were in good agreement with an error percentage of ±1% in the evaporator and ±7% in the condenser.

Yingshuang Wang et al [5] experimentally investigated heat transfer performance of shell and tube heat exchanger with segmental baffle and flower baffle. They maintained same inlet conditions on both arrangements. The overall performance of flower baffle is 20–30% more efficient than the segmental baffle. The flow resistance and heat transfer correlations were developed. He mentioned that the design of a shell and tube heat exchanger is better for the segmental to the flower baffle arrangement and give the best thermal performance.

Srbislav B. Genic et.al [6] conducted an experiment to calculate shell side heat transfer coefficient by using heat exchanger with three altered helical coil arrangements. In this analysis they used water as a working liquid and they also conducted the experiments. The Reynolds and Nusselt numbers were calculated by the shell side hydraulic diameter by them. They also stated that, the effect of heat transfer depend upon the geometrical constructions are axial pitch, radial pitch and winding angle of the helical coil of shell and coil heat exchanger and they also stated that shell side heat transfer coefficient mainly depends upon the hydraulic diameter of shell.

Luhong Zhang et.al. [7] Evaluated experimentally about the thermal and hydraulic performance of the shell and tube heat exchanger with one segmental and three continuous helical baffles (the helical angle of 70, 130 and 250). With their analysis we came to know that among all the four heat exchangers, both the shell side pressure drop and heat transfer rate is higher when helical angle equals 70, and the shell side heat transfer rate per unit pressure drop at this angle is the smallest. The concept of ‘liquid-flow distance’ was presented by them. They compared the relation between shell side Nusselt number and friction factor for the experimental models.

N. Jamshidi et al [8] conducted an experimental analysis on shell and coil heat exchanger for obtaining optimum heat transfer characteristics and design. They used Wilson plots to define the heat transfer coefficients in shell side and tube side. The geometrical parameters and the effect of liquid flow were calculated experimentally by them. They also used Taguchi method to investigate the heat transfer rate.

They mentioned that, to increase the coil diameter in tube side the overall Nusselt number and overall heat transfer coefficient are increases. He also quoted that on increasing the coil pitch it will affect the flow rate by decreasing the Nusselt number.

3.Catia Modeling

CATIA (Computer Aided Three-dimensional Interactive Application) is a multi-platform CAD/CAM/CAE commercial software suite developed by the French company Assault Systems. Written in the C++ programming language, CATIA is the cornerstone of the Assault Systems product lifecycle management software suite.

CATIA competes in the CAD/CAM/CAE market with Siemens NX, Pro/E, Autodesk Inventor, and Solid Edge as well as many others [7].

3.1MODELING:

In the process of the Catia modelling of Shell and Tube Heat Exchanger we have to design four Parts. They are,

- Tube Sheet
- Tubes
- Baffles
- Shell

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

3.1.1 TUBE SHEET:

Dimensions:

Diameter = 100mm
Pitch = 30mm
Hole diameter = 20mm

Used Catia Tools:

Circle, Rectangular Pattern, Circular Pattern, Pad, Pocket and Plane.

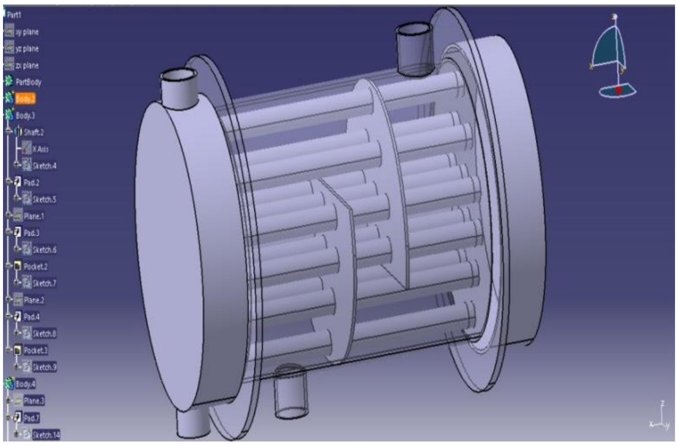


Fig.3.1 Designed Catia model of Tube Sheet

3.1.2 TUBES:

Dimensions:

Tube outer Diameter = 20mm
Thickness = 1mm
Tube Length = 600mm

Used Catia Tools:

Project 3D Elements and Pad.

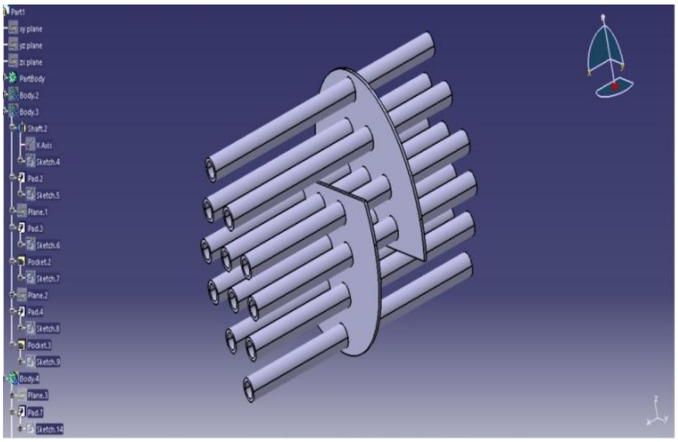


Fig 3.2 Designed Catia model of Tubes

3.1.3 BAFFLES:

1) Normal Baffle:

Dimensions:

Baffle Diameter = 90mm
Baffle thickness = 2mm
Baffle cut = 36%
Baffle spacing = 86mm
No. of Baffles = 6

Used Catia Tools:

Plane, Project 3D Elements, Pad, Pocket and Rectangular Pattern.

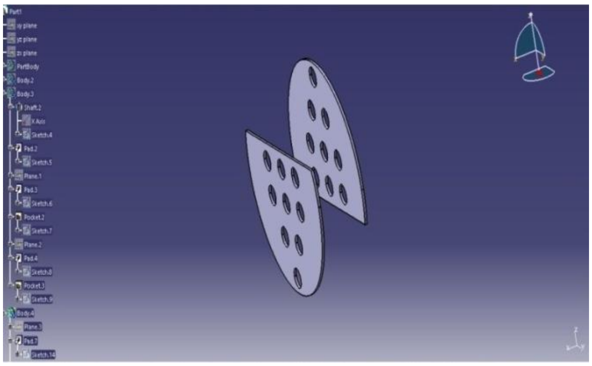


Fig 3.3 Designed Catia model of Normal Baffle

2) Quarter cut Baffle:

Dimension:

Baffle Inclination = 30^o
Baffle Diameter = 90mm
Baffle thickness = 2mm
Baffle cut = 25%
Baffle spacing = 86mm
No. of Baffles = 6

Used Catia Tools:

Plane, Project 3D Elements, Pad, Pocket and Rectangular Pattern.

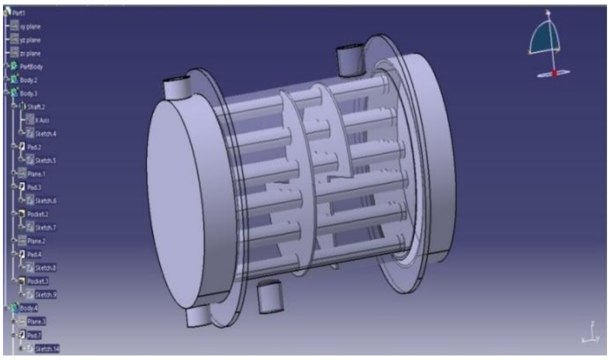


Fig 3.4 Quarter BC Shell and Tube Heat Exchanger

3) Mirror Quarter Baffle:

Dimensions:

Helix Diameter = 90mm
Helix Length = 600mm
Helical Pitch = 200mm
Baffle Thickness = 2mm

Used Catia Tools:

Helix, Point, Line, Rectangle and Pad.

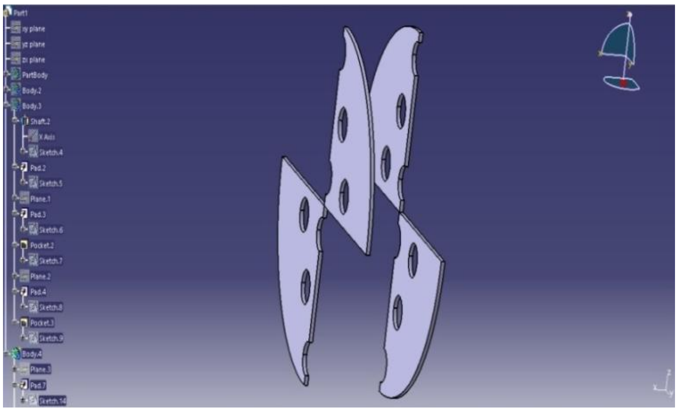


Fig 3.5 Designed Catia model of Mirror quarter baffles

3.1.4 SHELL:

Dimensions:

Shell inner Dia = 90mm
Shell Thickness = 5mm
Shell Length = 600mm

Used Catia Tools:

Project 3D Elements, Pad, Plane and Pocket.

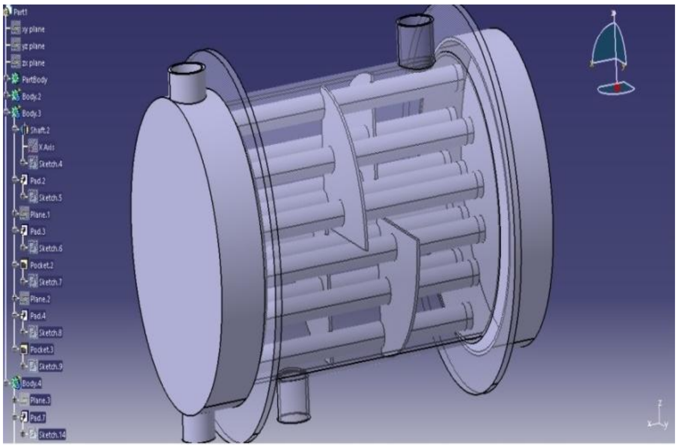


Fig 3.6 Designed Catia model of Shell

3.2 DESIGNED CATIA MODEL:

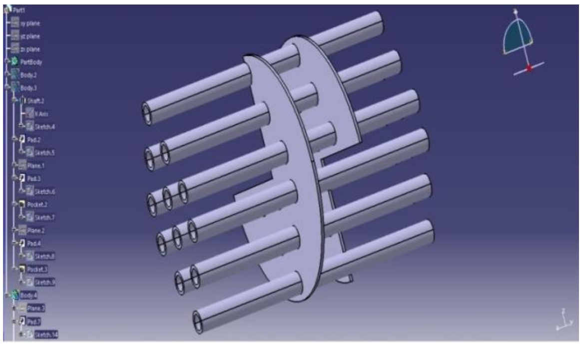


Fig 3.7 Designed Catia model of STHE with Baffle

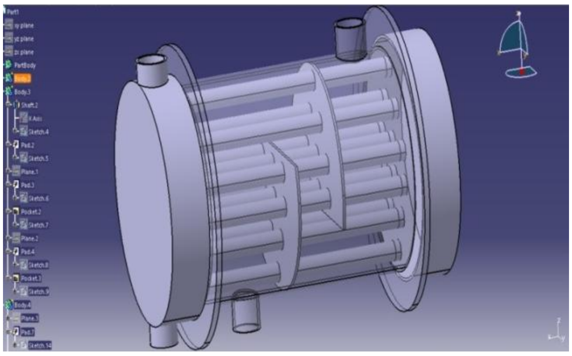


Fig.3.8 Designed Catia model of STHE with quarter cut Baffle

4. CFD ANALYSIS

Computational fluid dynamics (CFD) study of the system starts with the construction

4.1 GEOMETRY:

Heat exchanger is built in the ANSYS workbench design module [8]. It is a counter-flow heat exchanger. First, the fluid flow (fluent) module from the workbench is selected. The design modeler opens as a new window as the geometry is double clicked.

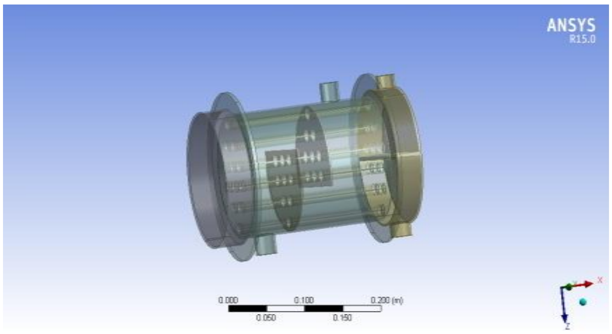


Fig. 4.1 Imported model in geometry

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

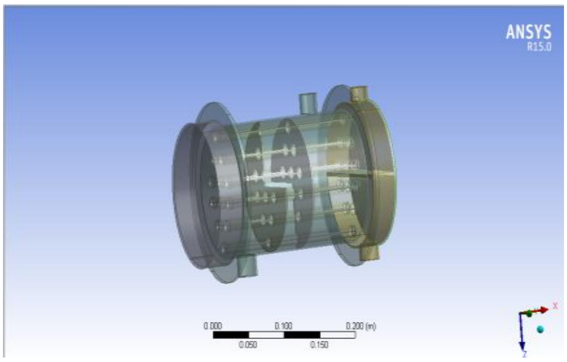


Fig. 4.2 shell side fluid domain in normal baffle

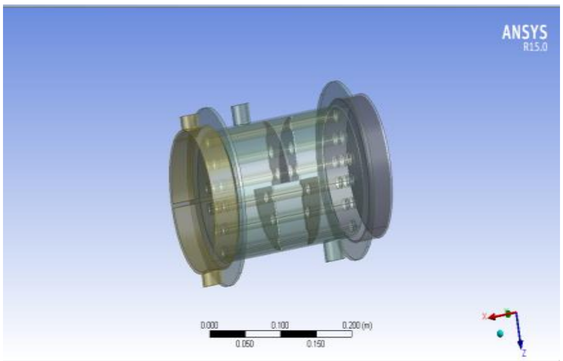


Fig. 4.3 shell side fluid domain in mirror cut baffle

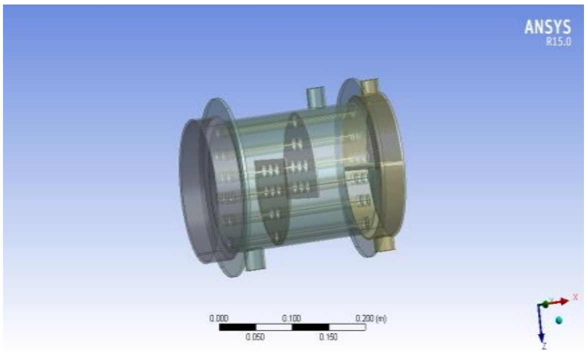


Fig. 4.4 shell side fluid domain in helical baffle

PART NUMBER	PART OF THE MODEL	STATE TYPE
1.	INNER FLUID	FLUID
2.	OUTER FLUID	FLUID
3.	BAFFLES(6)	SOLID
4.	SHELL(1)	SOLID
5.	TUBES(7)	SOLID
6.	TUBE SHEET(2)	SOLID

Table.4.1 geometry type and model

4.1.1 The Main Solver

The solver is the heart of CFD software [9]. It sets up the equation set according to the options chosen by the user and meshes points generated by the pre-processor, and solves them to compute the flow field. The process involves the following tasks:

- selecting appropriate physical model,
- defining material properties,
- prescribing boundary conditions,
- providing initial solutions,
- setting up solver controls,
- set up convergence criteria,
- solving equation set, and
- saving results

Once the model is completely set up, the solution starts and intermediate results can be monitored in real time from iteration to iteration. The progress of the solution process is displayed on the screen in terms of the residuals, a measure of the extent to which the governing equations are not satisfied.

4.2 MESHING

Initially a relativelycoarser mesh is generated. This mesh contains mixed cells (Tetra and Hexahedral cells) having both triangular and quadrilateral faces at the boundaries. Care is taken to use structured hexahedral cells as much as possible. It is meant to reduce numerical diffusion as much as possible by structuring the mesh in a well manner, particularly near the wall region [10. Later on, a fine mesh is generated. For this fine mesh, the edges and regions of high temperature and pressure gradients are finely meshed.

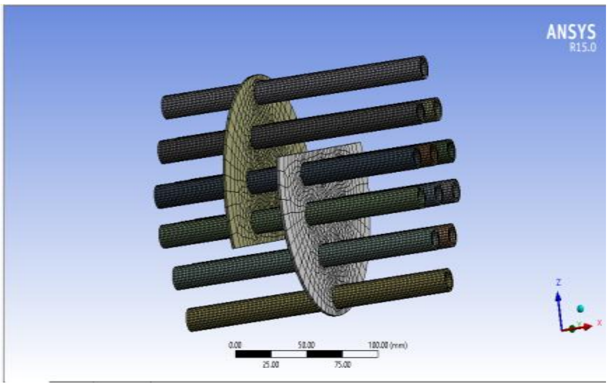


Fig 4.5 Shell and tube model after Meshing

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

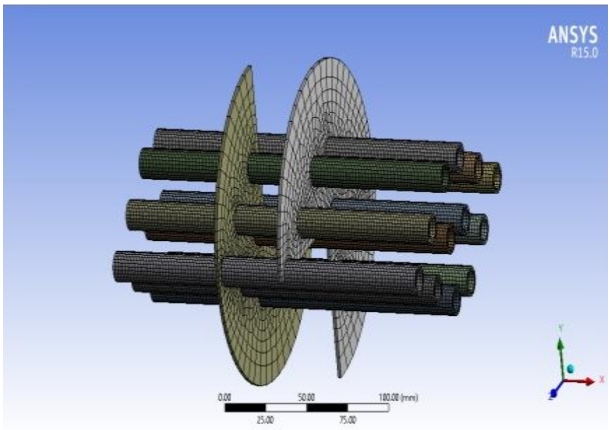


Fig 4.6 Shell and tube model of normal baffle after Meshing

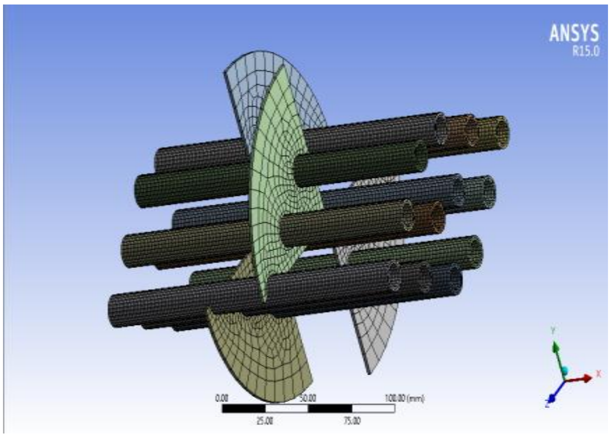


Fig 4.7 Shell and tube model of inclined baffle after Meshing

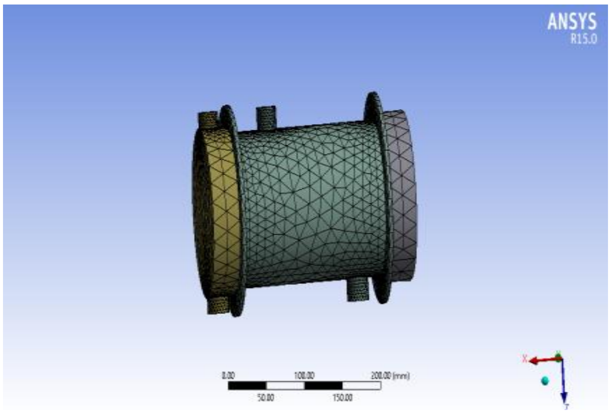


Fig 4.8 Shell and tube model after Meshing

The different surfaces of the solid are named as per required inlets and outlets for inner and outer fluids.

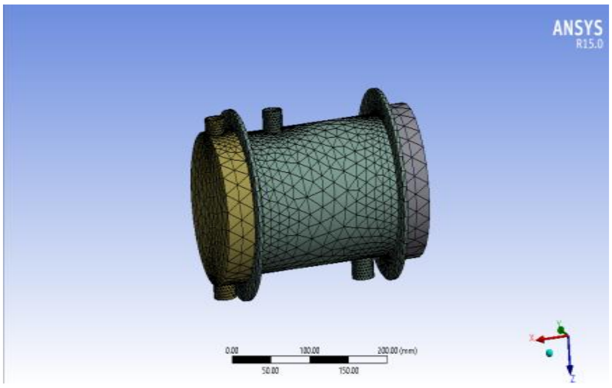


Fig 4.9 Meshed Model

Save project again at this point and close the window. Refresh and update project on the workbench. Now open the setup. The ANSYS Fluent Launcher will open in a window. Set dimension as 3D, option as Double Precision, processing as Serial type and hit OK. The Fluent window will open.

4.3 SETUP:

The mesh is checked and quality is obtained.

4.3.1 MATERIALS:

The create/edit option is clicked to add water-liquid, steel and copper to the list of fluid and solid respectively from the fluent database

4.3.2 CELL ZONE CONDITIONS: In cell zone conditions, we have to assign the conditions of the liquid and solid.

Table 4.2 cell zone conditions

Sno	PART/BODY	MATERIAL
1.	INNER FLUID	WATER-LIQUID
2.	OUTER FLUID	WATER-LIQUID
3.	TUBE SHEET	STEEL
4.	TUBES	COPPER
5.	BAFFLES	COPPER
6.	SHELL	STEEL

4.3.3 BOUNDARY CONDITIONS:

Boundary conditions are used according to the need of the model. The inlet and outlet conditions are defined as velocity inlet and pressure outlet. As this is a counter-flow with two tubes so there are two inlets and two outlets [7]. The walls are separately specified with respective boundary conditions. No slip condition is considered for each wall. Except the tube walls each wall is set to zero heat flux condition. The details about all boundary conditions can be seen in the table as given below.

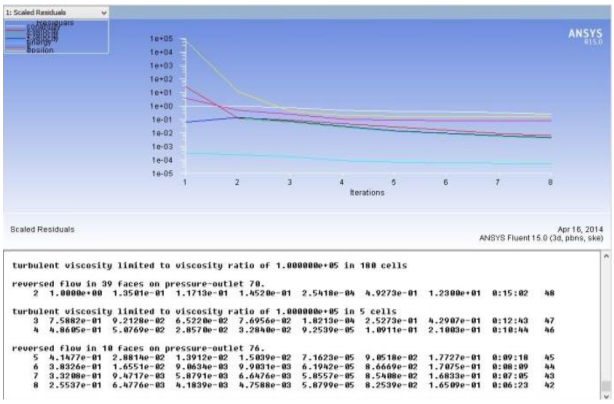
Table 4.3 boundary conditions

	BOUN DARY CONDITION TYPE	MASS FLOW RATE(kg/s)	TEMPER ATURE (k)
INNER INLET	Mass flow inlet	1	450
INNER OUTLET	Pressure outlet	-	-
OUTER INLET	Mass flow inlet	1	300
OUTER OUTLET	Pressure outlet	-	-

4.4 SOLUTION:

RUN CALCULATION:

After giving the boundary conditions to the inner and outer fluid, finally we have to run the calculations. The number of iteration is set to 500 and the solution is calculated and various contours, vectors and plots are obtained.



```
turbulent viscosity limited to viscosity ratio of 1.000000e+05 in 180 cells

reversed flow in 39 faces on pressure-outlet 70.
    2  1.0000e+00  1.3501e-01  1.1713e-01  1.4520e-01  2.5418e-04  4.9273e-01  1.2300e+01  0:15:02   48

turbulent viscosity limited to viscosity ratio of 1.000000e+05 in 5 cells
    3  7.5882e-01  9.2128e-02  6.5220e-02  7.6956e-02  1.8213e-04  2.5273e-01  4.2907e-01  0:12:43   47
    4  4.8605e-01  5.0769e-02  2.8570e-02  3.2840e-02  9.2539e-05  1.0911e-01  2.1003e-01  0:10:44   46

reversed flow in 10 faces on pressure-outlet 76.
    5  4.1477e-01  2.8814e-02  1.3912e-02  1.5039e-02  7.1623e-05  9.0518e-02  1.7727e-01  0:09:18   45
    6  3.8326e-01  1.6551e-02  9.0634e-03  9.9031e-03  6.1942e-05  8.6669e-02  1.7075e-01  0:08:09   44
    7  3.3208e-01  9.4717e-03  5.8791e-03  6.6476e-03  5.8557e-05  8.5408e-02  1.6833e-01  0:07:05   43
    8  2.5537e-01  6.4776e-03  4.1839e-03  4.7588e-03  5.8799e-05  8.2539e-02  1.6509e-01  0:06:23   42
```

Fig 4.10 Calculations was running

4.5The Post-processor

The post-processor is the last part of CFD software. It helps the user to examine the results and extract useful data. The results may be displayed as vector plots of velocities, contour plots of scalar variables such as pressure and temperature, streamlines and animation in case of unsteady simulation. Global parameters like drag coefficient, lift coefficient, Nusselt number and friction factor etc. may be computed through appropriate formulas. These data from a CFD post-processor can also be exported to visualization software for better display.

Several general-purpose CFD packages have been published in the past decade. Prominent among them are:PHOENICS,FLUENT, STAR-CD,CFX, CFD-ACE,ANSWER, CFD++, FLOW-3D and COMPACT. Most of them are based on the finite volume method. CFD packages have also been developed for special applications; FLOTHERM and ICEPAK for electronics cooling, CFX-TASCFLOW and FINE/TURBO for turbo machinery and ORCA for mixing process analysis are some examples. Most CFD software packages contain their own grid generators and post processors. Software such as ICEM CFD, Some popular visualization software used with CFD packages are TECPLOT and FIELDVIEW.

4.6 OVERVIEW OF FLUENT PACKAGE

FLUENT is a state-of-the-art computer program for modeling fluid flow and heat transfer in complex geometries. FLUENT provides complete mesh flexibility, solving your flow problems with unstructured meshes that can be generated about complex geometries with relative ease. Supported mesh types include 2D triangular/quadrilateral, 3D FLUENT also allows user to refine or coarsen grid based on the flow solution.

FLUENT is written in the C computer language and makes full use of the flexibility and power offered by the language. Consequently, true dynamic memory allocation, efficient data structures, and flexible solver control are all made possible. In addition, FLUENT uses a client/server architecture, which allows it to run as separate simultaneous processes on client desktop workstations and powerful compute servers, for efficient execution, interactive control, and complete flexibility of machine or operating system type. All functions required to compute a solution and display the results are accessible in FLUENT through an interactive, menu-driven interface. The user interface is written in a language called Scheme, a dialect of LISP. The advanced user can customize and enhance the interface by writing menu macros and functions.

5. RESULTS AND DISCUSSIONS

The objective of this project is to evaluate the maximum pressure drop in shell side of Shell and Tube Heat Exchanger. When pressure difference is more, it will caused for increasing of entering velocity and discharge. Baffles are the main parameters which influence the pressure drop. These Baffles are optimized by using fluid flow analysis.

The Effect of Staggered Baffle Arrangement over Pressure Drop in Shell and Tube Heat Exchanger

The baffle member plays an important role in STHE and it supports tube bundle and equally distribute the fluid in shell side. When segmental baffles are used in STHE which have many disadvantages .The low heat transfer is achieved due to the flow stagnation i.e., dead zones which are created at the corners between baffle and shell wall . It requires higher pumping power and it creates high pressure drop under the same heat load. The heat transfer rate, velocity and pressure drop in shell side are calculated below

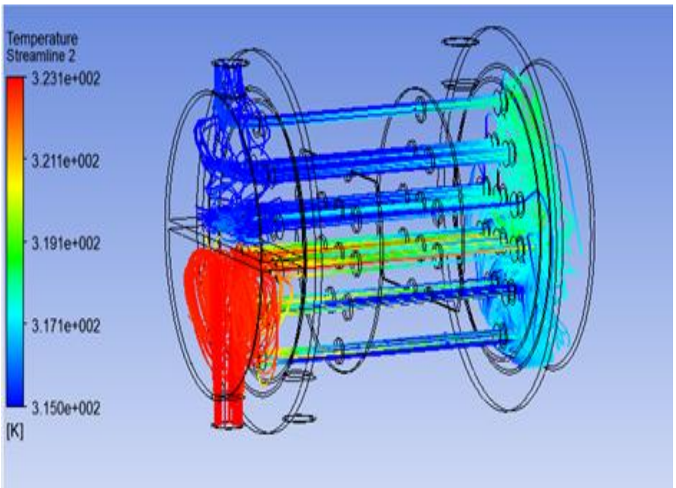


Fig 5.1 Tube side heat transfer of 20% glycerin

5.1 REPORTS:

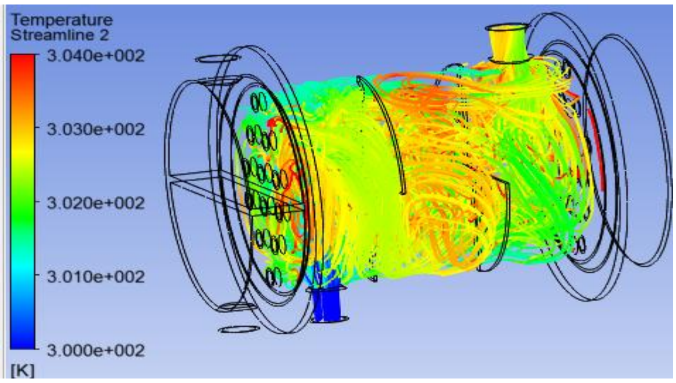


Fig 5.2 shell side heat transfer of 20% glycerin

5.2 Normal Baffle

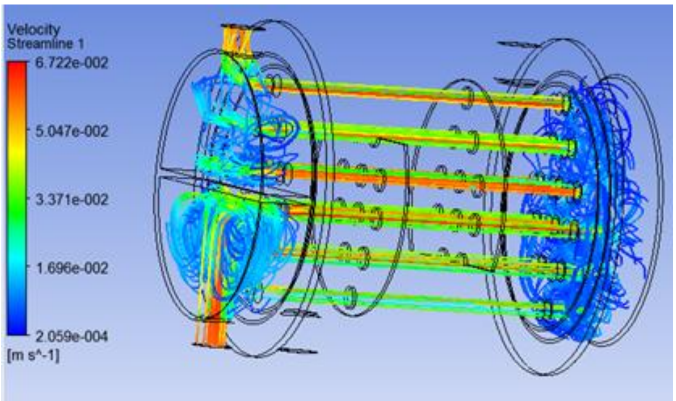


Fig 5.3 shell side heat transfer of 20% glycerin

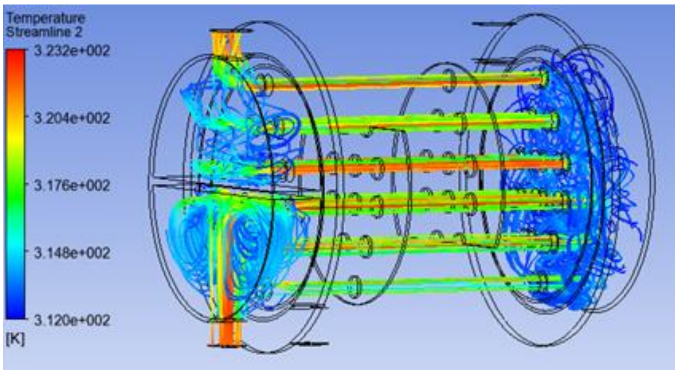


Fig 5.4 shell side heat transfer of 20% glycerin

From the above two figures we can clearly notice that the change in temperature in the shell side as well as tube side of the shell and tube heat exchanger and the change in temperature is shown in the form of streamlines and the temperature difference is due to heat transfer action takes place between the liquid in the tube side to the liquid in the shell side with the help of conduction and convection modes of heat transfer.

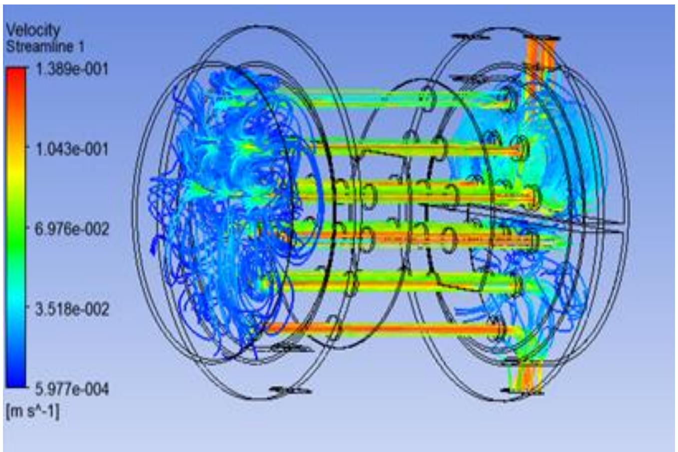


Fig 5.5 Velocity variation 40% glycerin

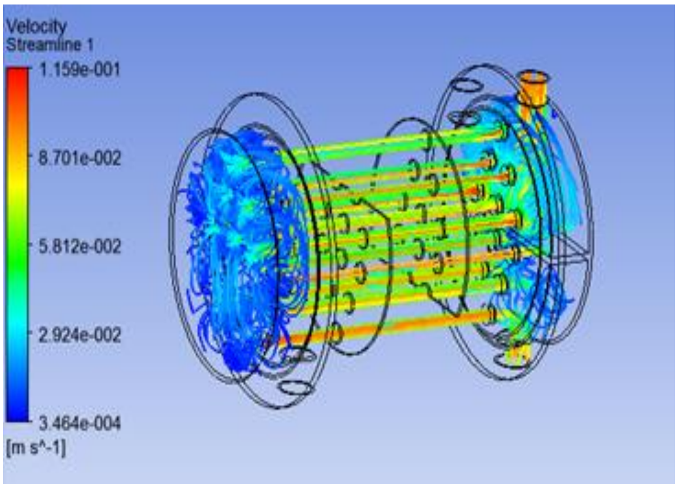


Fig 5.6 Pressure variation in 40% glycerin

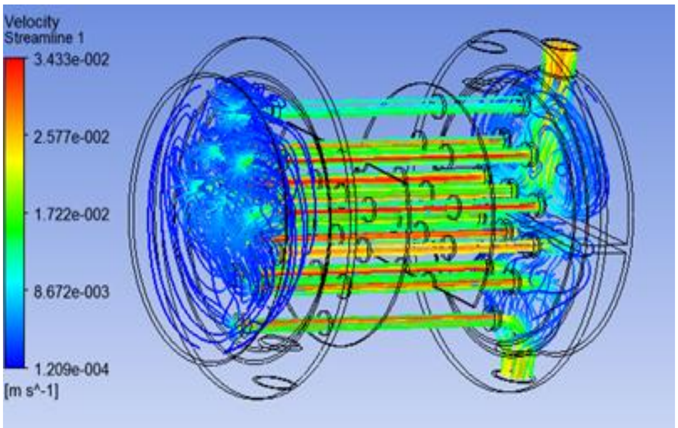


Fig 5.7 Pressure variation in 60% glycerin

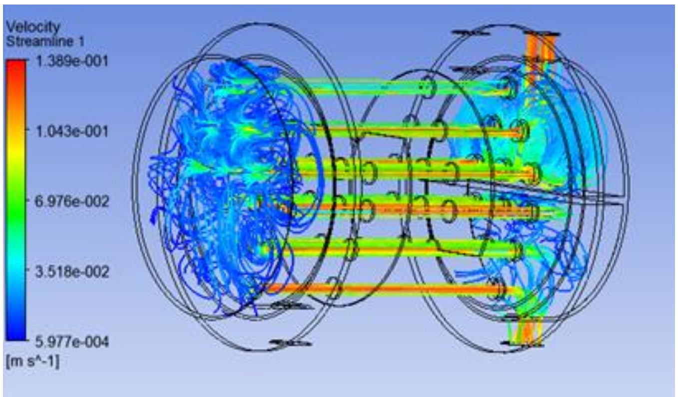


Fig 5.8 Velocity variation in 60% glycerin

5.3 Glycerin at 80% proportion:

In the case of glycerin content the direction of flow of cold fluid changes in X-Y plane and also in Z direction. As a result the total length of flow increased and heat transfer rate also proportionately increase. The heat transfer rate increases up to 8% when the 20% glycerin is replaced by 100% water content and it is up to 20% when 40% glycerin is replaced by 20% glycerin.

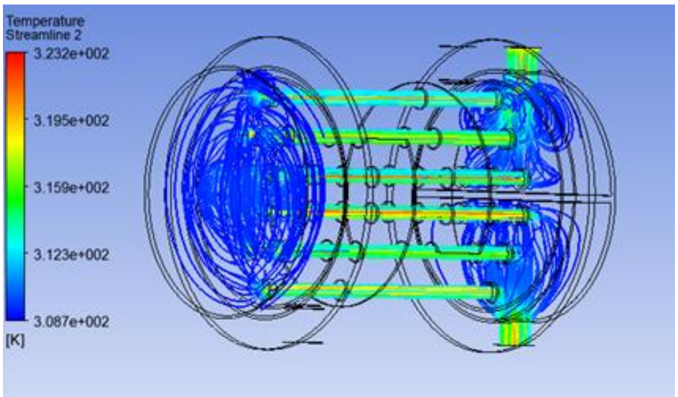


Fig 5.9 Temperature variation in 80% glycerin

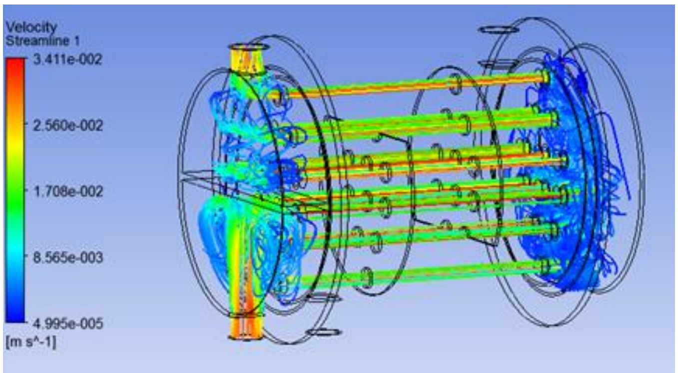


Fig 5.10 Velocity variation in 80% glycerin

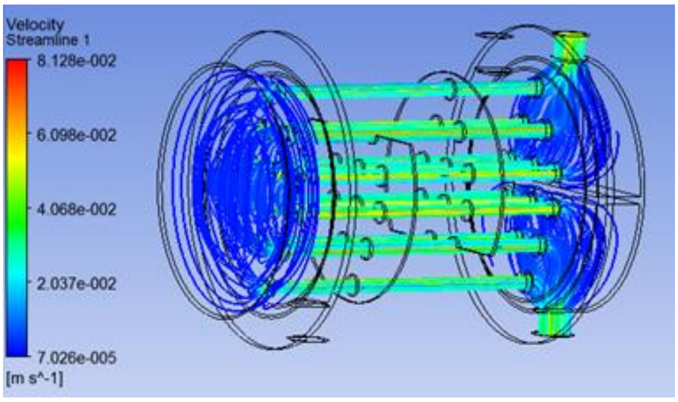


Fig 5.11 Velocity variation in 100% Glycerin

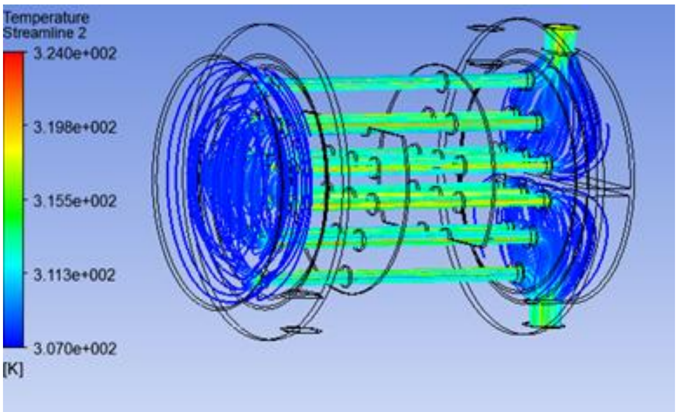


Fig 5.12 Temperature variation in 100% Glycerin

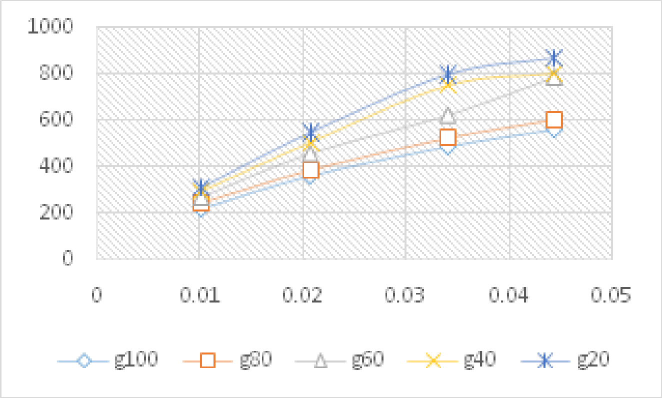


Fig 5.13 Temperature variation graph.

The influence of pressure drop along tube side of shell and tube heat exchanger with different concentrations of Nano fluids is calculated. The temperature drop for water to water transformation is 12% from 100% glycerin and 16% from 80% glycerin and 20% from 60% glycerin and 26% from 40% glycerin and finally 30% from 20% glycerin.

Generally, higher the velocity usually the higher the heat transfer coefficient. The higher the velocity, the higher the pressure drop. In other words, the higher the velocities and pressure drops, the more effective the transfer is. For a given transfer, the greater the flow rate, the higher the pressure drop. Usuallyin shell side of the heat exchanger which drastically changes the behavior of fluid which may leads to change in mass flow rate.

6. CONCLUSION

The performance of a shell and tube heat exchanger mainly depends on different design modifications in shell side as well as tube side. So further numerical analysis is conducted on circular and elliptical tube geometry models of the single shell and multiple tube heat exchanger with 25%, quarter, mirror quarter baffle cuts at 450, 600 and 900 tube orientations and strip inserts are considered to enhance the heat transfer rate by minimizing the pressure drop.

In the present study numerical analysis is carried out on the heat transfer behavior of Nano fluids through STHE under turbulent flow conditions with various percentages. water with different glycerin concentrations are used to enhance the heat transfer rate by minimizing the pressure drop. Based on the simulation results it is concluded that by different percentages of the glycerin the heat transfer rates were observed. By comparing the results glycerin of 20% with water 80% content will give the best results. And at different flow rates the heat transfer rate is increased by 12% compared as compared to the remaining percentages of the water.

7. REFERENCES

[1]. Joydeep Barman, A.K. Ghoshal. “Performance analysis of finned tube and un baffled shell-and-tube heat exchangers”, International Journal of Thermal Sciences 46 (2007) 1311–1317.

[2]. M.R. Salimpour, “Heat transfer coefficients of shell and coiled tube heat exchangers”, Experimental Thermal and Liquid Science 33 (2009) 203–207.

[3]. Jian-Fei Zhang, Bin Li, Wen-Jiang Huang, Yong-Gang Lei, Ya-Ling He, Wen-Quan Tao, “Experimental performance comparison of shellside heat transfer for shell-and-tube heat exchangers with middle overlapped helical baffles and segmental baffles.” ChemicalEngineering Science 64 (2009)1643-1653.

[4]. F. Vera-Garcia, J.R. García-Cascales, J. Gonzalvez-Macia, R.Cabello, R. Llopis, D. Sanchez , E. Torrella, “A simplified model for shell-and-tubes heat exchangers: Practical application”, Applied Thermal Engineering 30 (2010) 1231–1241.

[5]. Yingshuang Wang, Zhichun Liu, Suyi Huang, Wei Liu, Weiwei Li, “Experimental investigation of shell-and-tube heat exchanger with a new type of baffles”, Heat Mass Transfer (2011) 47:833–839.

[6]. Srbislav B. Genic, Branislav M. Jacimovic, Marko S. Jaric, Nikola J. Budimir , Mirko M. Dobrnjac, “Research on the shell-side thermal performances of heat exchangers with helical tube coils”, International Journal of Heat and Mass Transfer 55 (2012) 4295-4300.

[7]. Luhong Zhang, Youmei Xia, Bin Jiang, Xiaoming Xiao, Xiaoling Yanga, “Pilot experimental study on shell and tube heat exchangers with small-angles helical baffles”, Chemical Engineering and Processing 69 (2013) 112– 118.

[8]. N. Jamshidi, M. Farhadi, D.D. Ganji, K. Sedighi, “Experimental analysis of heat transfer enhancement in shell and helical tube heat exchangers”. Applied Thermal Engineering 51 (2013) 644-652.

[9]. N. Jamshidi, M. Farhadi, D.D.Ganji, K.Sedighi, “Experimental analysis of heat transfer enhancement in shell and helical tube heat exchanger”, Applied Thermal Engineering 51 (2013) 644-652.

[10]. Xiang-hui Tan, Dong-sheng Zhu, Guo-yan Zhou, Li-ding Zeng,“Heat transfer and pressure drop performance of twisted oval tube heat exchanger”, Applied Thermal Engineering 50 (2013) 374-383.