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

Fluent Analysis of Shell and Tube Heat Exchanger with Different Materials and Fluids

H. Vasudeva Rao
Department of Mechanical Engineering,
Sistam Engineering College,
Srikakulam, Andhra Pradesh-532404, India.

Dr. J. Bala Bhaskara Rao
Department of Mechanical Engineering,
Sistam Engineering College,
Srikakulam, Andhra Pradesh-532404, India

ABSTRACT:

Heat transfer is considered as transfer of thermal energy from physical body to another. Heat transfer is the most important parameter to be measured as the performance and efficiency of the double pipe heat exchanger. By using CFD simulation software, it can reduce the time and operation cost compared to experimental calculations, in order to measure the optimum parameter and the behavior of this type of heat exchanger.

The area of application is used is Food industry, Beverage industry, Bio-process industry. The aim of this study is to assess the influence of the Double pipe heat exchanger to get best heat transfer by using Double pipe heat exchanger. The purpose of this study is to use CFD software and Experimental setup to analyze the Temperature drop, Pressure drop and Friction factors by varying under different inlet conditions like Temperature and Flow rate as a function of both inlet velocity and temperature variations and changing heat exchanger tube material properties like copper and aluminum.

In this experiment, heat transfer from hot fluid to cold fluid by Double pipe heat exchanger is experimentally investigated by using different organic fluids like Benzene, Glycol, Transformer oil, Acetone and Water to get better heat transfer, the same thing is validated in CFD analysis. The experiment is performed at Laminar flow and Turbulent flow under different flow arrangements like Parallel flow and Counter flow. Same flow variations are validated in CFD analysis.

The experiment is carried out by introducing baffle segmental above the inner tube at different angular orientations i.e., 0º, 45º, 90º and 135º performed in both laminar flow and turbulent flow to get better heat transfer rate. The study is heat transfer coefficient of Double pipe heat exchanger under various working fluids with different operating conditions, it is predicted that Counter flow exhibits better heat transfer than Parallel flow.

Key words: Heat exchangers, cfd, catia, Double pipe heat exchanger, baffles.

1. INTRODUCTION:

1.1 Heat Exchanger:

Heat exchanger is a device, such as an automobile radiator, used to transfer heat from a fluid on one side of a barrier to a fluid on the other side without bringing the fluid into direct contact (Fogiel, 1999). Usually, this barrier is made from metal which has good thermal conductivity in order to transfer heat effectively from one fluid to another fluid. Besides that, heat exchanger can be defined as any of several devices that transfer heat from a hot to a cold fluid [1]. In engineering practical, generally, the hot fluid is needed to cool by the cold fluid. Moreover, heat exchanger is defined as a device used to exchange heat from one medium to another often through metal walls, usually to extract heat from a medium flowing between two surfaces.

Cite this article as: H.Vasudeva Rao & Dr. J.Bala Bhaskara Rao, "Fluent Analysis of Shell and Tube Heat Exchanger with Different Materials and Fluids", International Journal & Magazine of Engineering, Technology, Management and Research, Volume 5, Issue 2, 2018, Page 107-117.

1.2 Double pipe heat exchanger:

A typical double-pipe heat exchanger consists of one pipe placed concentrically in side another of larger diameter with appropriate fittings to direct the flow from one section to the next, as shown in figure (1.2). Double-pipe heat exchangers can be arranged in various series and parallel arrangements to meet pressure drop and mean temperature difference requirements. The major use of double-pipes exchangers is for sensible heating or cooling of process fluids where small heat transfer areas are required [2]. This configuration is also very suitable. When one or both fluids is at high pressure. The major disadvantage is that double-pipe heat exchangers are bulky and expensive per unit transfer surface. Inner tube being may be single tube or multi-tubes Fig.(1.3). If heat transfer coefficient is poor in annulus, axially finned inner tube (or tubes) can be used. Double-pipe heat exchangers are built in modular concept, i.e., in the form of hair fins.

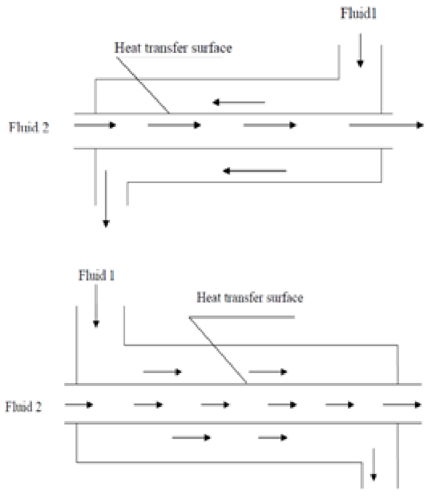


Fig 1.1: Double pipe heat exchanger

1.3 Heat Transfer Considerations:

Heat transfer is the term used for transfer of thermal energy from a hot to a colder body. Theoretically on a microscopic scale, thermal energy is related to the kinetic energy of molecules [3]. The greater a material's temperature, the greater the thermal agitation of its constituent molecules. Then the regions containing greater molecular kinetic energy will pass this energy to regions with less kinetic energy. So when a physical body likes an object or fluid, is at a different temperature than its surroundings or another body, heat transfer will occurs in such a way that the body and the surroundings reach thermal equilibrium. Heat transfer always occurs from a hot body to a cold one, a result of the second law of thermodynamics [4]. Where there is a temperature difference between objects in proximity, heat transfer between them can never be stopped but can only be slowed down.

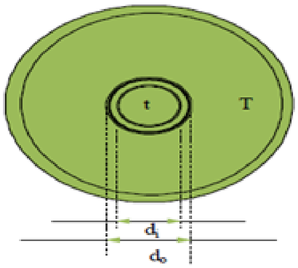


Fig 1.2: End view of a tubular heat exchanger

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

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.

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 [5]. 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.

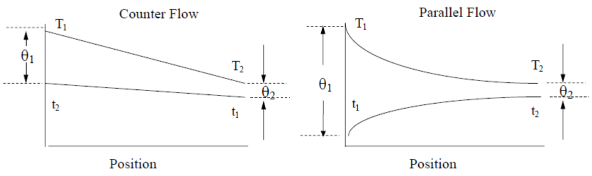


Fig.1.11:Log Mean Temperature Differences

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:

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

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 [6]. Another interesting observation from the above Figure is that counter flow is more appropriate for maximum energy recovery.

2. LITERATURE REVIEW

Muhammad Mahmoud Salam Butta, Nasir Hayat et, al [1] "CFD Applications in Various Heat Exchangers Design." A Review, Department of Mechanical Engineering, University of Engineering & Technology, Applied Thermal Engineering, 2011.It has been found that CFD employed for the fluid flow mal-distribution, fouling, pressure drop and thermal analysis in the design and optimization phase. CFD provides cost effective alternative, speedy solution and eliminate the need of prototype. The simulations results ranging from 2% to 10% with the experimental studies. In some exceptional cases, it varies to 36%.

J.S. Jayakumar et, al [2] "Helically Coiled Heat Exchangers." 2008, Chemical Engineering Research and Design 221-232. Observed that the use of constant values for the transfer and thermal properties of the fluid resulted in inaccurate heat transfer coefficients. The fluid flow conditions in the outer pipe had a major contribution on the overall heat transfer coefficient. The study showed that during the design of a double pipe helical heat exchanger, the design of the outer pipe should get the highest priority in order to get a highest overall heat transfer coefficient

S.M. Mahajani, J.C. Mandal, et, al [3] "Experimental and CFD estimation of heat transfer in helically coiled heat exchangers." 2008, Chemical Engineering Research and Design 221-232. Studied the constant thermal and transport properties of the heat transfer medium and their effect on the prediction of heat transfer coefficients. An experimental setup was made for studying the heat transfer and also CFD was used for the simulation of the heat transfer. Based on both the experimental and simulation results a correlation was established for the inner heat transfer coefficient.

Chandra Sekhara Reddy. et, al [4] "Experimental investigation of heat transfer coefficient and friction factor of ethylene glycol water based TiO2 Nano fluid in double pipe heat exchanger with and without helical coil inserts." International Communications in Heat and Mass Transfer, Volume 50, January 2014, Pages 68-76.Have investigated the heat transfer coefficient and friction factor of TiO2 Nano fluid having volume concentration range from 0.0004% to 0.02% in a base fluid having 40% of ethylene glycol and 60% of distilled water, flowing in a double pipe heat exchanger with and without helical coil inserts.

Topakoglu et, al [5]"Steady laminar flows of an incompressible viscous fluid in curved pipes" Journal of Mathematics and Mechanics.Used an approximate solution using stream-functions to determine the flow pattern for steady laminar flows of an incompressible viscous fluid in curved pipes. Results showed that the flow rate depended on two independent variables, the Reynolds number and the curvature of the pipe.

Ender Ozden et, al [6] "Shell Side CFD Analysis of a Small Shell and Tube Heat Exchanger." Middle East Technical University, 2010. Has investigated the design of shell and tube heat exchanger by numerically modeling in particular the baffle spacing, baffle cut and shell diameter dependencies of heat transfer coefficient and pressure drop.

Prof. Naresh B. Dhamane et al[7] "Heat Transfer Analysis of Helical Strip Insert with Regularly Spaced Cut Sections Placed inside a Circular Pipe." International Journal of Modern Engineering Research (IJMER) Vol. 2, Issue. 5, Sep.-Oct 2012 pp-3711-3716. Presents an experimental study of heat transfer and friction characteristics in turbulent flow generated by a helical strip inserts with regularly spaced cut passages, placed inside a circular pipe across the test section. The experiments were conducted for water flow rates in the range of Re 5000 to Re 30000.

W. A. Aly et, al [8] "Numerical study on turbulent heat transfer and pressure drop of Nano fluid in coiled tube in-tube heat exchangers." Energy Conversion and Management, Volume 79, March 2014, Pages 304-316. Has carried out computational fluid dynamics (CFD) analysis to study the heat transfer and pressure drop characteristics of water-based Al2O3 Nano fluid flowing inside coiled tube-in-tube heat exchangers. He found that the heat transfer coefficient increases by increasing the coil diameter and nano particles volume concentration.

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.

3.1 PROBLEM DESCRIPTION AND MODELLING

The experiment is carried under different flow arrangements i.e. turbulent flow and laminar flow which are further carried out in parallel and counter flow using different organic fluids like glycerin, glycol, acetone, benzene, transformer oil, water keeping water as base fluid and by varying only hot fluids [8].

The inlet flow conditions like mass flow rate are varied maintaining the temperature constant i.e. at inner inlet 312^{o}K and at outer inlet 300^{o}K, the mass flow rates in laminar flow for inner inlet& outer inlet is taken as 0.0133, 0.0166, 0.02 kg/s and 0.01, 0.0133, 0.0166 kg/s similarly in turbulent flow the mass flow rates for inner inlet & outer inlet are taken as 1.2, 1.8, 2.4 kg/s&1.166, 1.66&2.166 kg/s, the Physical parameters like Materials properties and baffle segmental were introduced into the double pipe heat exchanger fluent model as the properties cannot be varied in experimental setup heat exchanger [8].

Materials with good thermal conductivities like copper and aluminum where opted for analyzing the heat exchanger. The baffle segmental are introduced to the heat exchanger on the surface of inner pipe at different angular orientations i.e. 0^o, 45^o, 90^o and 135^o to predict that at which angular orientation better pressure drop, and heat transfer co-efficient are building up.

3.2 Experimental setup

Setup:

Fig 3.1 Double pipe heat exchanger

MODELING:

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

Without Baffle-segmental

1. Inner pipe
2. Outer pipe

-With Baffle-segmental

1. Inner pipe
2. Outer pipe
3. Baffle segmental

3.2.1 Inner pipe:

Dimensions:

Pipe outer Diameter = 25mm
Pipe inner Diameter = 21mm
Thickness = 4mm
Tube Length = 1400mm

Used Catia Tools:

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

3.2.2 TUBES:

Dimensions:

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

Used Catia Tools:

Project 3D Elements and Pad.

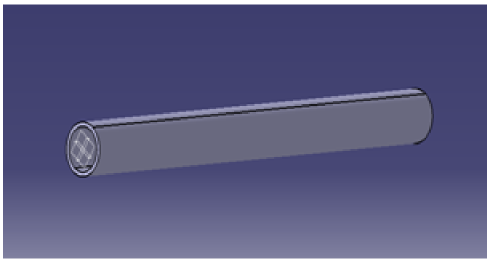


Fig 3.2 Designed Catia model of Inner pipe

Outer pipe:

Dimensions:

Pipe outer Diameter = 38mm
Pipe inner Diameter = 32mm
Thickness = 6mm
Tube Length = 1100mm

Used Catia Tools:

Circle, Pad, Plane and Pocket

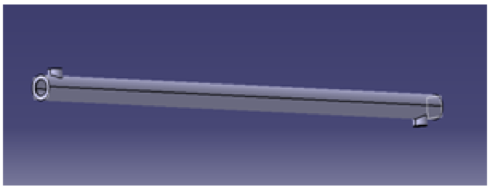


Fig 3.3 Designed Catia model of Outer pipe

2) Baffle-segmental:

Dimensions:

Baffle Diameter = 28mm
Baffle thickness = 2mm
Baffle cut = 36%
Baffle spacing = 220mm
No. of Baffles = 8

Used Catia Tools:

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

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

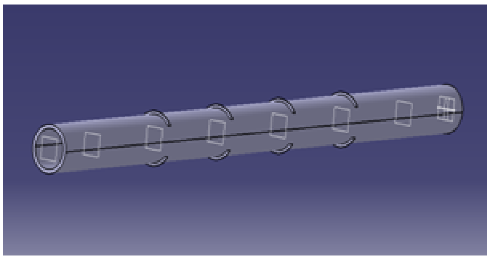


Fig 3.4 Designed Catia model of Baffle

3) Helical 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 Helical Baffle

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.

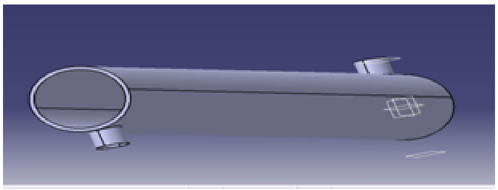


Fig 3.6 Designed Catia model of Shell

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.

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.

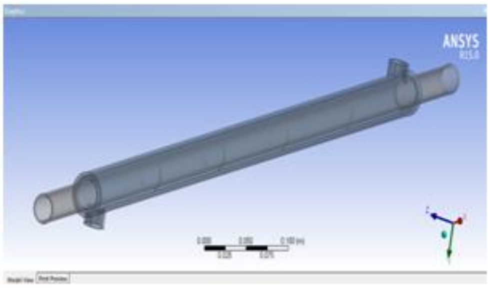
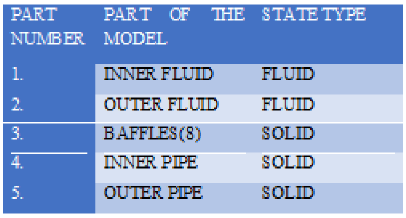


Fig. 4.1 Imported model in geometry

4.1.1 The Main Solver:

The solver is the heart of CFD software. 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 [9]. The process involves the following tasks:

Table.4.1 geometry type and model



PART NUMBER	PART OF THE MODEL	STATE TYPE
1.	INNER FLUID	FLUID
2.	OUTER FLUID	FLUID
3.	BAFFLES(8)	SOLID
4.	INNER PIPE	SOLID
5.	OUTER PIPE	SOLID

- 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.

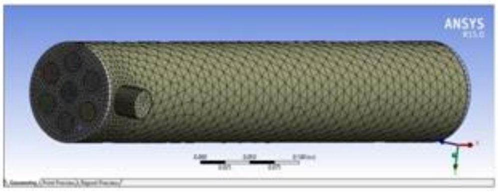


Fig 4.2 Shell and tube model after Meshing

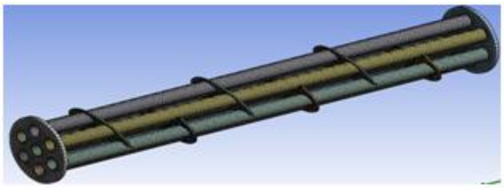


Fig 4.3 Shell and tube model of inclined baffle after Meshing

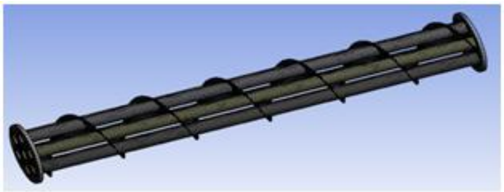


Fig 4.4 Shell and tube model of helical after Meshing

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

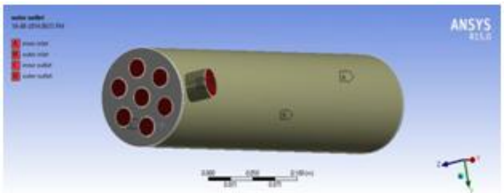


Fig 4.5 Named selections

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.

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

	BOUNDARY CONDITION TYPE	MASS FLOW RATE (kg/s)	TEMPERATURE (k)
INNER INLET	Mass flow inlet	0.02	312
OUTER INLET	Mass flow inlet	0.01	300

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.

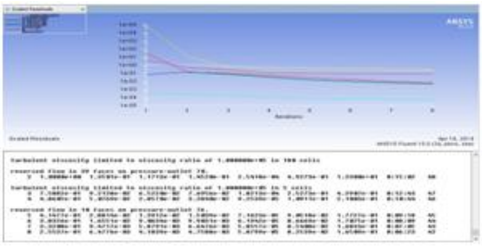


Fig 4.6 Calculations was running

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

5.1 REPORTS:

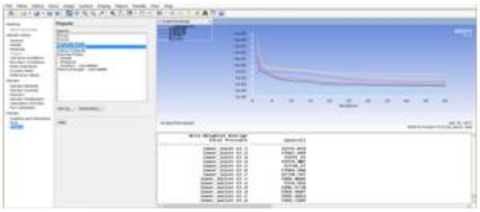


Fig 5.1 verifying the reports.

5.2 Normal Baffle

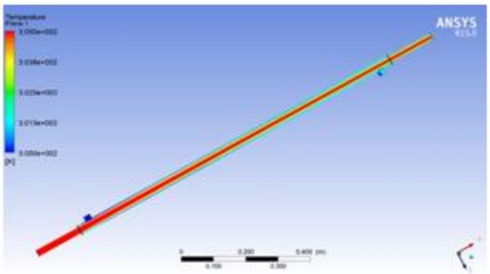


Fig 5.3 Temperature variation in normal baffle

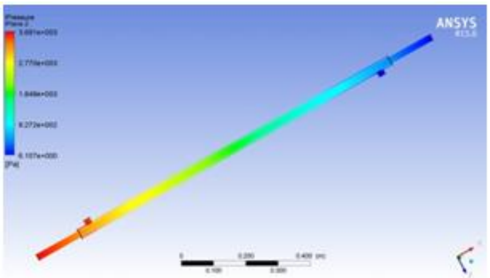


Fig 5.4 Pressure variation

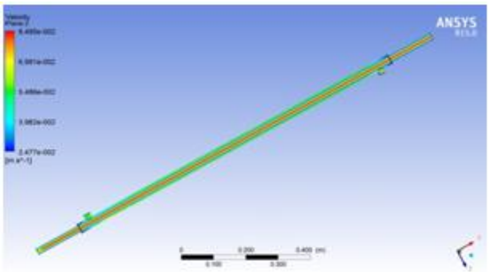


Fig 5.5 Velocity variation

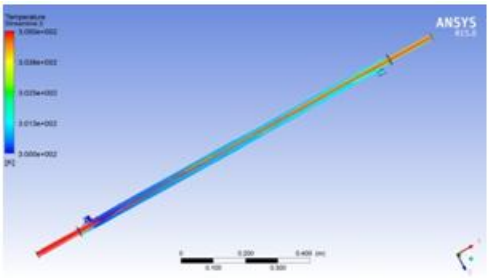


Fig 5.6 Temperature variation

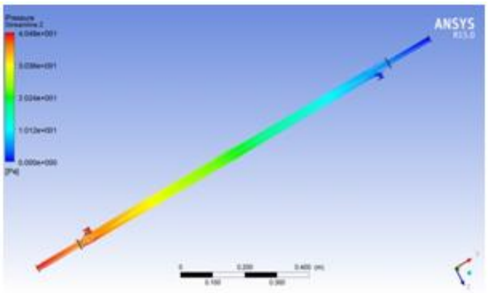


Fig 5.7 Pressure variation

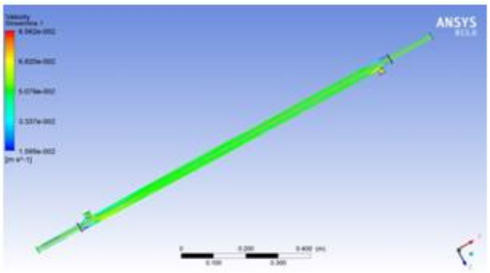


Fig 5.8 Velocity variation

5.3 AT 0.02 MASS FLOW:

Staggered baffles are replaced by inclined baffles and thermal properties are calculated. Due to inclination of the baffle it reduces the stagnation points and dead zones. The flow path increased in shell side of shell and tube heat exchanger with these inclined baffles.

The flow pattern and pressure drop are calculated and tabulated. As compare to normal baffles these baffles gives less pressure drop in shell side of shell and tube heat exchanger.

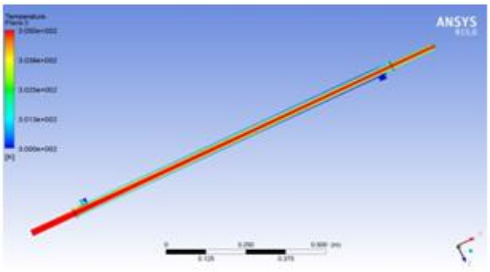


Fig 5.9 Temperature Variation

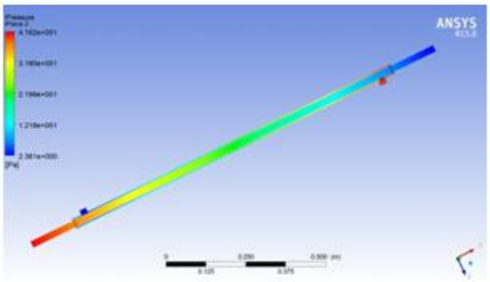
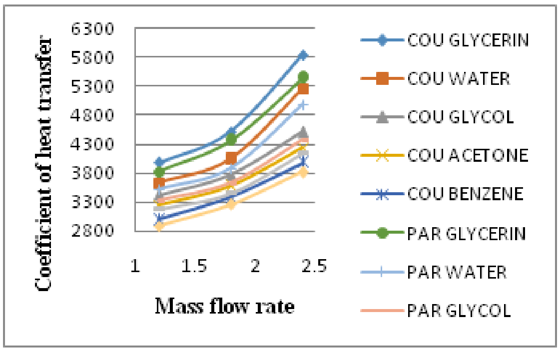


Fig 5.10 Velocity variation in inclined baffle.

In this project we are calculate Glycerin for aluminum material acquired better heat transfer co-efficient followed by glycerin for copper material compared to other organic fluids on increasing mass flow rate. From the figure 5.53 that heat transfer co-efficient is directly proportional to mass flow rate



i.e. by increasing mass flow rate, the heat transfer co-efficient increases.

6. CONCLUSION:

At first, the main objective of this project is to create a validation between Experimental &CFD results which was performed using different organic solutions like Glycerine, Ethylene glycol, Acetone, Benzene and Transformer oil to check the percentage error, in order to affirm the experimental setup. As the percentage of error is within 6% the experimental setup is validated.

In Parallel flow arrangement, considering laminar flow. At a flow rate of 0.02 m/s by using Glycerine fluid acquired better heat transfer of 275.666 watts.

In counter flow arrangement, considering laminar flow. At a flow rate of 0.02 m/s by using Glycerine fluid acquired better heat transfer of 285.1023842 watts. Hence, Glycerine fluid in counter flow arrangement is better effective.

Now the whole experiment is performed using CFD analysis in turbulent flow arrangement, which is further classified into parallel & counter flow for different fluids and materials like copper and aluminium. Aluminium exhibited better Co-efficient of heat transfer (Q_A) than that of copper, along with the fluid glycerin at different mass flow rates in both counter flow & parallel flow arrangement's, so it is concluded that Double pipe heat exchanger using Aluminium material with glycerin as fluid is more efficient compared to copper material & other fluids.

In Aluminium tube material, Parallel flow arrangement considering turbulent flow. At a flow rate of 2.4 m/s by using Glycerine fluid acquired better heat transfer of 4942.201 watts. In Aluminium tube material, counter flow arrangement considering turbulent flow. At a flow rate of 2.4 m/s by using Glycerine fluid acquired better heat transfer of 5693.850461watts. Hence, Glycerine fluid in counter flow arrangement is better effective. Further the experiment is carried out by introducing baffle segmental at different angular orientations like 0º, 45º, 90º & 135º above the surface of the inner pipe of Double pipe heat exchanger using Aluminium material. Since the Aluminium material is affirmed to be more efficient than copper.

The experiment is performed using CFD analysis in laminar and turbulent flow arrangement, which is further classified into parallel & counter flow for Glycerine fluid. In Aluminium tube material inserted with Baffle segmental at an angle of 45^o counter flow arrangement considering laminar flow. At a flow rate of 0.02 m/s by using Glycerine fluid acquired better heat transfer of 8494.884watts. In Aluminium tube material inserted with Baffle segmental at an angle of 45^o counter flow arrangement considering turbulent flow.

At a flow rate of 2.4 m/s by using Glycerine fluid acquired better heat transfer of 11821.26542watts. Hence, The Aluminium tube material inserted with Baffle segmental at an angle of 45^o counter flow arrangement considering turbulent flow. At a flow rate of 2.4 m/s by using Glycerine fluid is better effective. The study results resolved that the double pipe heat exchanger with baffle segmental at 45 degree angular orientation using glycerin as working fluid give better heat transfer.

7. REFERENCES:

[1] Muhammad Mahmoud Salam Butta, Nasir Hayat et.al,CFD Applications In Various Heat Exchangers Design: A Review, Department Of Mechanical Engineering, University Of Engineering & Technology, Applied Thermal Engineering, 2011.

[2] J.S.Jayakumar, Helically Coiled Heat Exchangers Chemical Engineering Research and Design 221-232.

[3] S.M. Mahajani, J.C. Mandal, Experimental and CFD estimation of heat transfer in helically coiled heat exchangers. 2008, Chemical Engineering Research and Design 221-232.

[4] M. Chandra Sekhara Reddy and Veeredhi Vasudeva Rao, Experimental investigation of heat transfer coefficient and friction factor of ethylene glycol water based TiO2 Nano fluid in double pipe heat exchanger with and without helical coil inserts, International Communications in Heat and Mass Transfer, Volume 50, January 2014, Pages 68-76.

[5] Topakoglu, Steady laminar flows of an incompressible viscous fluid in curved pipes. Journal of Mathematics and Mechanics, Vol. 16(12):1321-1337.

[6] Ender Ozden, Ilker Tari, Shell Side CFD Analysis of a Small Shell and Tube Heat Exchanger, Middle East Technical University, 2010.

[7] Prof. Naresh B. Dhamane et al,Heat Transfer Analysis of Helical Strip Insert with Regularly Spaced Cut Sections Placed inside a Circular Pipe. International Journal of Modern Engineering Research (IJMER) Vol. 2, Issue. 5, Sep.-Oct. 2012 pp-3711-3716.

[8] W. A. Aly, Numerical study on turbulent heat transfer and pressure drop of Nano fluid in coiled tube in-tube heat exchangers. Energy Conversion and Management, Volume 79, March 2014, Pages 304-316.

[9] Liu. L and et. al, The Potential Impact of Nano fluid Enhancements on the Performance of Heat Exchangers. Heat Transfer Engineering, Volume 33, Issue 1, 2012, pages 31-41.

[10] G. E. Kondhalkar & V. N. Kapatkat, Performance Analysis ofSpiral tube heat exchanger used in oil extraction system. International Journal of Modern Engineering Research, Vol.2, Issue.3, PP – 930 – 936, ISSN No.2249 – 6645, May – June2012.