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

CFD ANALYSIS OF DOUBLE PIPE HEAT EXCHANGER WITH DIFFERENT BAFFLE SEGMENTS

S.Venkata Sai Praveen[1], Dr.S.M.JameelBasha[2], Dr.B.Narayana Reddy[3] & V.Chengal Reddy[4]
[1].M.Tech Student,CAD/CAM,Chadalawada Ramanamma Engineering College, Tirupati, Andhra Pradesh.
[2].Professor,CREC,Chadalawada Ramanamma Engineering College, Tirupati, Andhra Pradesh.
[3].Associate Professor, Chadalawada Ramanamma Engineering College, Tirupati, Andhra Pradesh.
[4].Assistant Professor, CREC,Chadalawada Ramanamma Engineering College, Tirupati, Andhra Pradesh.

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 behaviour 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 analyse the Temperature drop, Pressure drop 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 aluminium.

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 carried out by introducing baffle segmental above the inner tube at different angular orientations i.e., 0º, 45º, 90º and 135º performed 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].

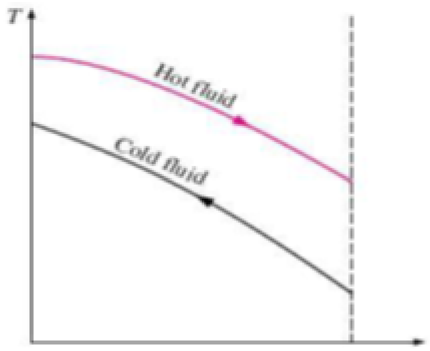
Cite this article as: S.Venkata Sai Praveen, Dr.S.M.JameelBasha, Dr.B.Narayana Reddy & V.Chengal Reddy., "CFD Analysis Of Double Pipe Heat Exchanger With Different Baffle Segments", International Journal & Magazine of Engineering, Technology, Management and Research (IJMETMR), ISSN 2348-4845, Volume 9 Issue 7, July 2022, Page 1-15.

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

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.

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.



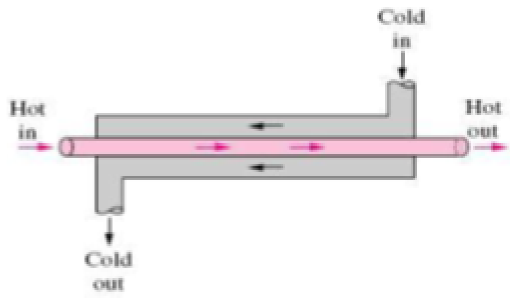


Fig.1.1 Double pipe Heat exchanger

1.2 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 1.11, the temperature difference will vary along the length of the HX, and this must be taken into account in the analysis.

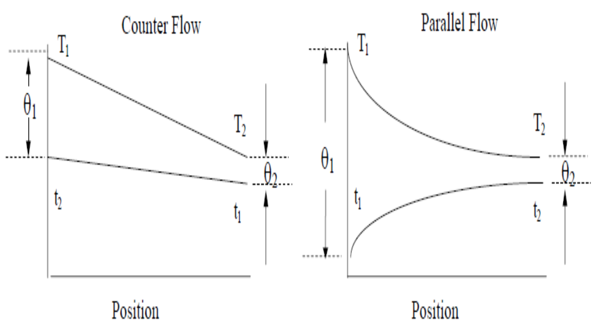


Fig.1.2 Log Mean Temperature Differences

1.3: 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 $\Delta\theta$ 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 Hayatet, 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 TiO_2 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 TiO_2 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

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

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 Experimental setup

Fig 3.1 Double pipe heat exchanger

3.2 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

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

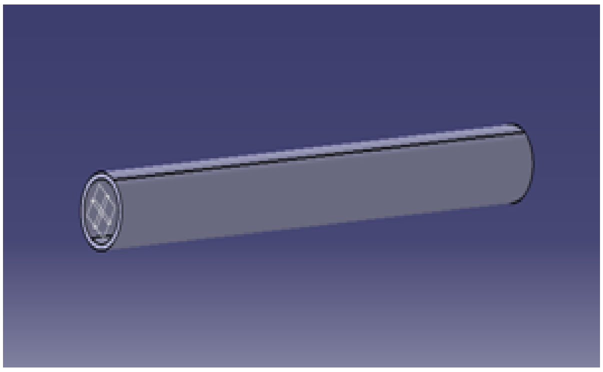
3.2.1 Inner pipe:
Dimensions:
Pipe outer Diameter = 25mm
Pipe inner Diameter = 21mm
Thickness = 4mm
Tube Length = 1400mm
Used Catia Tools:
Circle, Rectangular Pattern, Circular Pattern, Pad, Pocket and Plane.

3.2.2 TUBES:
Dimensions:
Tube outer Diameter = 20mm
Thickness = 1mm
Tube Length = 600mm
Used Catia Tools:
Project 3D Elements and Pad.
Outer pipe:
Dimensions:
Pipe outer Diameter = 38mm
Pipe inner Diameter = 32mm
Thickness = 6mm
Tube Length = 1100mm
Used Catia Tools:
Circle, Pad, Plane and Pocket

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.

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

3.2.3 SHELL:
Dimensions:
Shell inner Dia = 90mm
Shell Thickness = 5mm
Shell Length = 600mm
Used Catia Tools:
Project 3D Elements, Pad, Plane and Pocket.



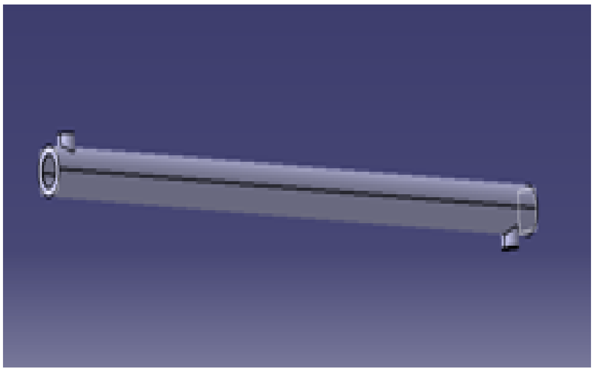
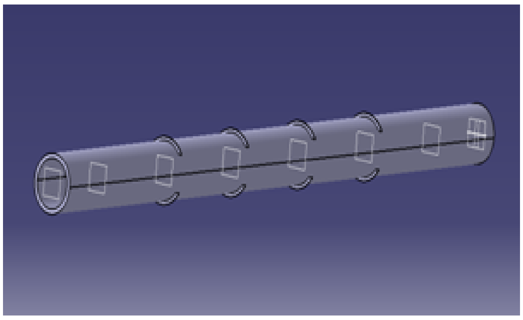


Fig 3.2 Designed Catia model of Inner pipe
Fig 3.3 Designed Catia model of Outer pipe

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



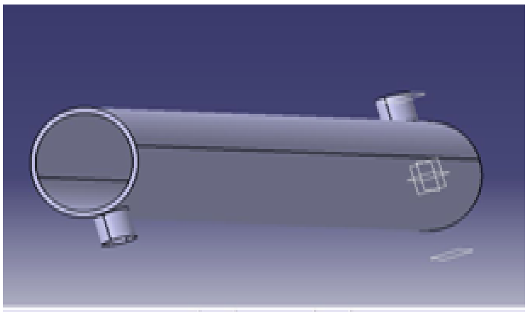


Fig 3.4 Designed Catia model of Baffle
Fig 3.5 Designed Catia model of Shell

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.

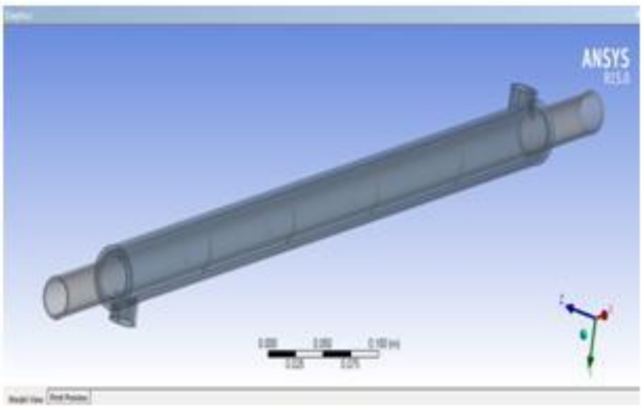


Fig. 4.1 Imported model in geometry

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

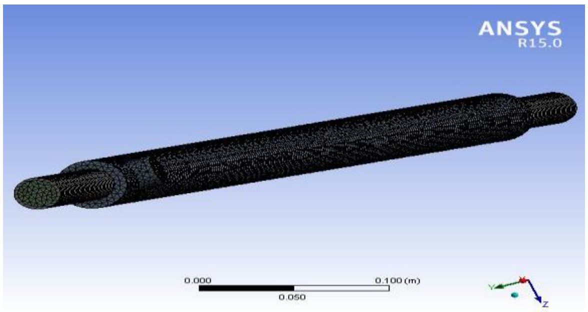
PART NUMBER	PART OF THE MODEL	STATE TYPE
1.	INNER FLUID	FLUID
2.	OUTER FLUID	FLUID
3.	BAFFLES(8)	SOLID
4.	INNER PIPE	SOLID
5.	OUTER PIPE	SOLID

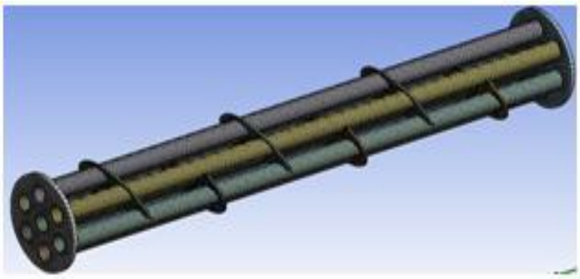
- selecting appropriate physical model,
- defining material properties,
- prescribing boundary conditions,
- providing initial solutions,
- setting up solver controls,
- set up convergence criteria,
- solving equation set, and
- saving results

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 relatively coarser 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.





4.2 Double pipe model after Meshing

Fig 4.3 Shell and tube model of inclined baffle after Meshing

The different surfaces of the solid are named as per required inlets and outlets for inner and outer fluids.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.	INNER PIPE	COPPER
4.	OUTER PIPE	COPPER
5.	BAFFLES	COPPER

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.

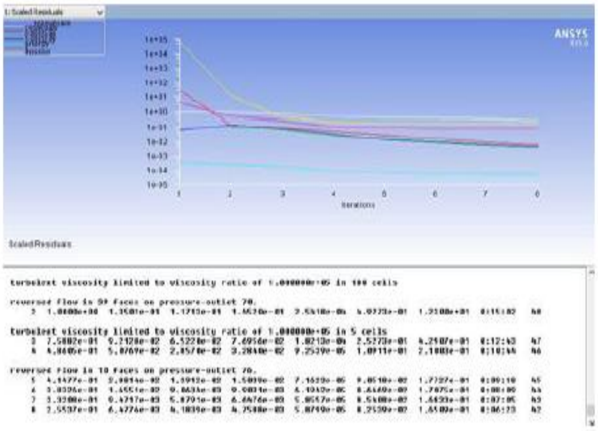


Fig 4.5 Calculations was running

4.5 The 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 postprocessor 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, CFXTASCFLOW 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.

5. RESULTS AND DISCUSSIONS

The objective of this project is to predict that at which condition the maximum heat transfer rate is obtaining in double pipe Heat Exchanger by introducing different organic solutions like glycerin, glycol, acetone, benzene, water liquid & by changing parameters like inlet flow conditions, flow arrangements i.e. counter flow & parallel flow which are further divided into laminar flow & turbulent flow by maintaining constant temperature at inner inlet and outer inlet. Different materials like copper & aluminum were used by introducing baffle-segmental on the surface of inner pipe at different angular orientations i.e. 0º,45º, 90º, 135º. Baffles are the main parameters which influence the pressure drop & heat transfer co-efficient. These Baffles are optimized by using fluid flow analysis.

5.1 Defining Material Properties

Material properties were derived from tables based on the temperature which was being calculated in the model. The material was defined in FLUENT using its material browser. For the different flow arrangement problem model certain properties were defined by the user prior to computing the model, these properties were: thermal conductivity, density, heat capacity at constant pressure, ratio of specific heats, and dynamic viscosity.

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

Material properties Different material properties	Density (ρ) kg/m3	Thermal conductivity(K) W/mk	Specific heat CP j/kgK
Copper	8978	387.6	381
Aluminium	2719	203.2	871

5.2 Defining fluid Properties

Water was used as the base fluid flowing through outer pipe. Fluids like glycerin, glycol, acetone, benzene & water are used as hot fluids which are allowed to flow through inner pipe by maintaining constant inner temperature. The fluids were defined in FLUENT using its fluids browser. For the different flow arrangement problem model certain properties were defined by the user prior to computing the model, these properties were: thermal conductivity, density, viscosity, specific heat.

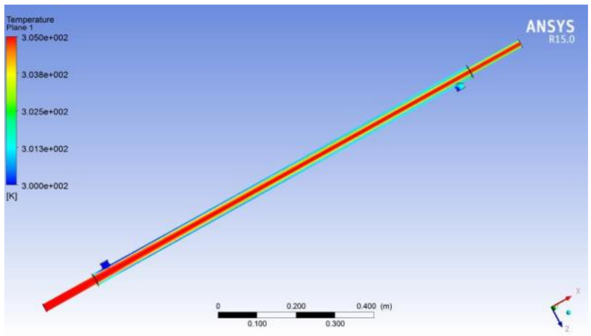
FLUIDS	DENSITY (Kg/m^3)	VISCOSITY (Ns/m^2)	SP. HEAT (Kj/kg k)	THERMALCONDUCTIVITY (W/mk)
ACETONE	791	0.00033	2160	0.180
BENZENE	876.5	0.00058	1821	0.167
GLYCERIN	1261	0.799	2813	0.285
GLYCOL	1116	0.0157	2200	0.258
WATER	998.2	0.001003	4174	0.6

5.3 Experimental validation for parallel flow:

The temperature, pressure & velocity variation in a parallel flow double pipe heat exchanger of copper material performed for laminar flow is as shown in below profiles.

5.3.1Temperature, pressure & Velocity Profile for parallel flow Heat Exchanger:

At mass flow rate 0.02 (Plane representation)



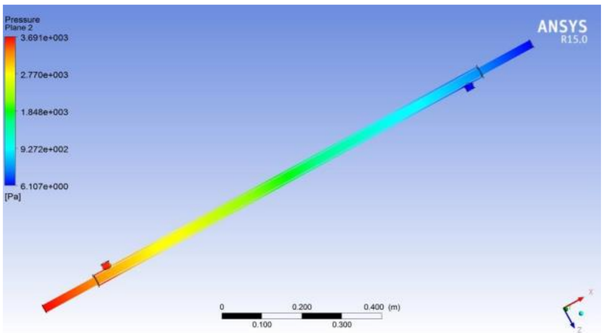


Fig 5.1 Temperature variation

Fig 5.2 Pressure variation

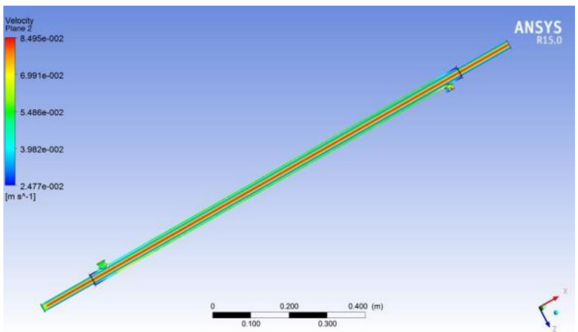
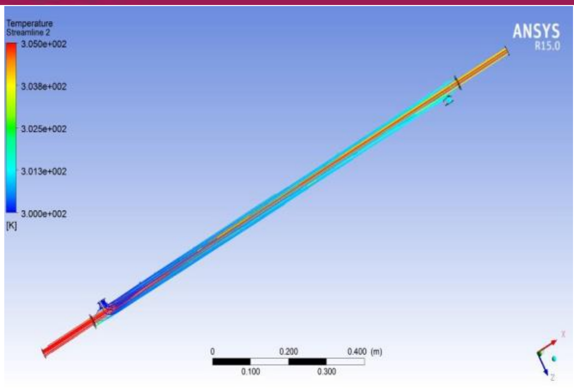


Fig 5.3 Velocity variation

-At mass flow rate 0.02 (Stream line representation)

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



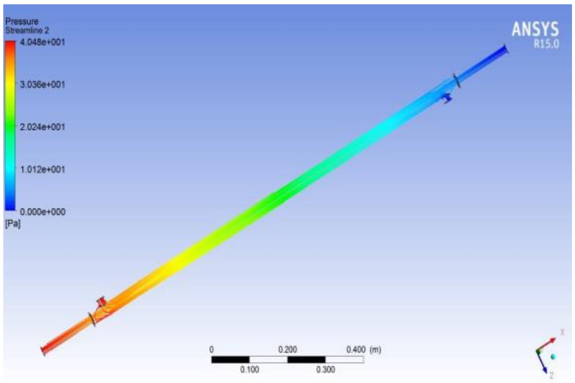


Fig 5.4 Temperature variation

Fig 5.5 Pressure variation

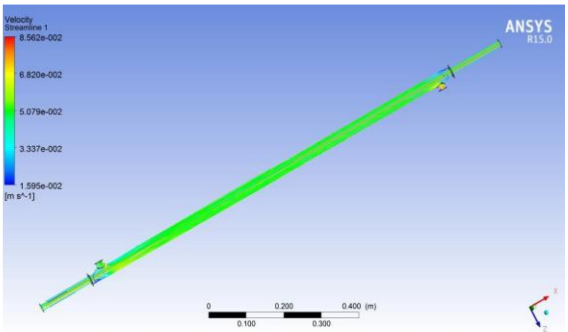


Fig 5.6 Velocity variation

5.3.2 Sample Calculation for Parallel Flow:

Thi= 312 K Tci= 300 K

Tho=307.138 K Tco= 303.597 K

Mh = (1.2)/60 = 0.02 litres/s

Mc = (1)/60 = 0.016667 litres/s

Cph = 2813 J/kgk Cpc = 4174 J/kgk

Qh= Mh Cph (Thi –Tho) watts

= 273.53612 Watts

Qc= Mc Cpc (Tco– Tci) watts

= 250.245213 Watts

Qa = (Qh + Qc)/2

= 261.8906667Watts

LMTD = (θ2 – θ1)/ Ln (θ2/θ1)

θ2 = Thi– Tci = 12 K

θ1 = Tho – Tco = 3.5408 K

LMTD = 6.93062357 K

Overall heat transfer coefficient:

Uo=Qa/ (Ao* LMTD) = (261.890667) / (0.0863*6.93062357) =391.4060045 W/m^2K

5.3.3 Parallel flow study results:

For parallel flow heat exchangers the hotter fluid will lower in temperature as it loses heat to the cooler fluid which will then rise in temperature due to the heat transfer.

Figure 5.7 shows this gradual temperature change in both flow paths. This is the correct curve form already proven for co-current flow heat exchangers.

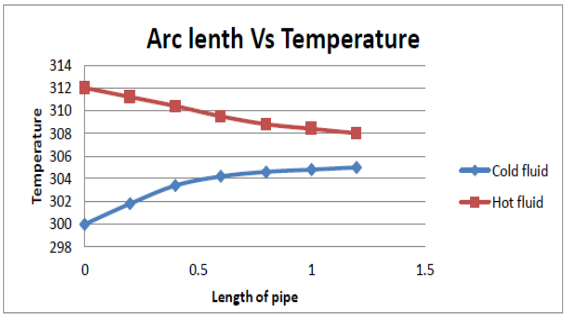
From the figure 5.8, the hot fluid inlet and outlet temperatures is directly proportional to mass flow rate i.e. on increasing the mass flow rate the temperature of hot fluid increases. The graph illustrates that glycerine has acquired highest hot inlet & outlet temperatures compared to remaining fluids

From the figure 5.9, the cold fluid inlet and outlet temperatures is inversely proportional to mass flow rate i.e. on increasing the mass flow rate the temperature of cold fluid decreases.

From the figure 5.10 shows that co-efficient of heat transfer directly proportional to mass flow rate i.e. by increasing mass flow rate, the co-efficient of heat transfer increases. As shown in figure, Glycerine CFD acquired better co-

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

efficient of heat transfer compared to other organic fluids on increasing mass flow rate.



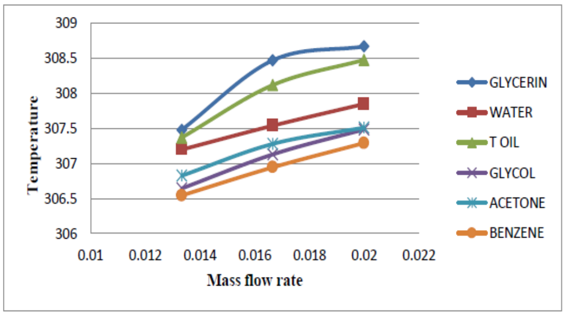
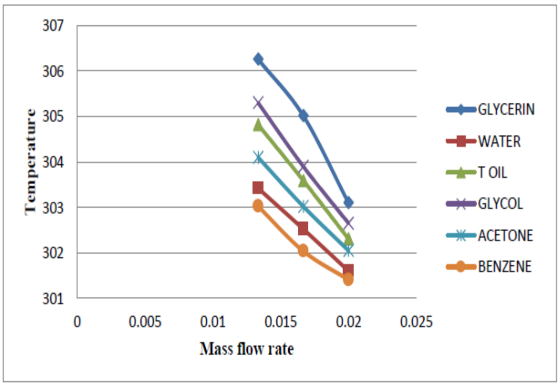


Fig 5.7 Parallel Flow profile curve
Fig 5.8 Hot Fluid Temperature vs. Mass flow rate



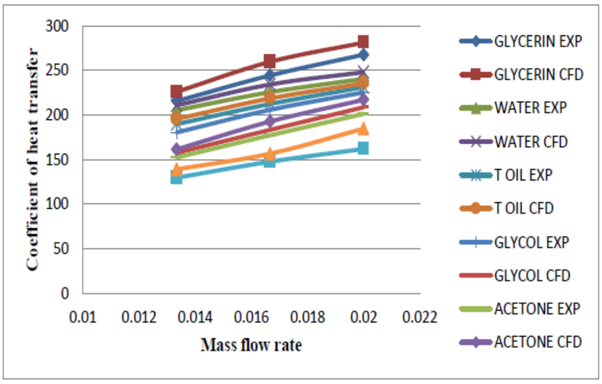


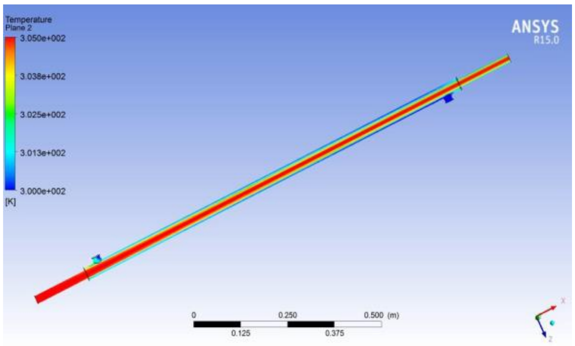
Fig 5.9Cold Fluid Temperature vs.Mass flow rate Fig 5.10Co-efficient of heat transfer vs. Mass flow rate

5.4 Experimental validation for counter flow:

5.4.1 Laminar Flow in a Counter Heat Exchanger FLUENT model:

Temperature, pressure & Velocity Profile for counter flow Heat Exchanger:

At mass flow rate 0.02 (Plane representation)



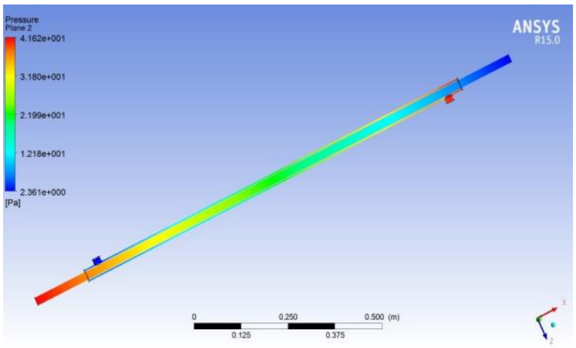


Fig 5.11 Temperature variation
Fig 5.12 Pressure variation

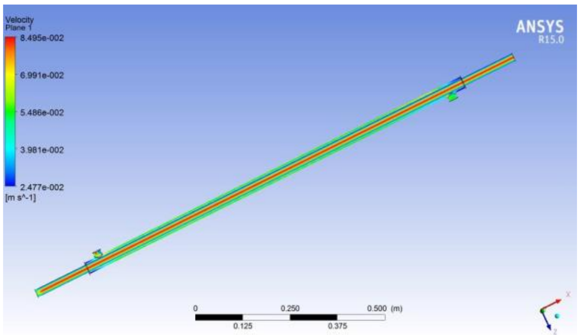
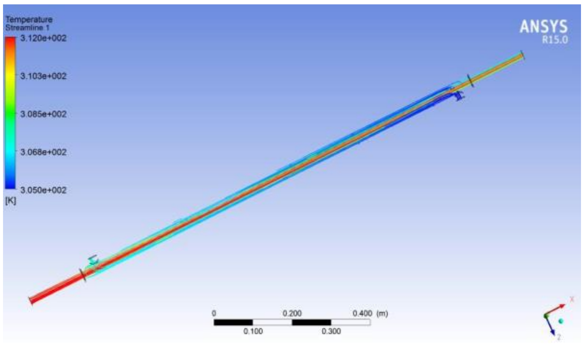


Fig 5.13 Velocity variation

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

-At mass flow rate 0.02 (Stream line representation)



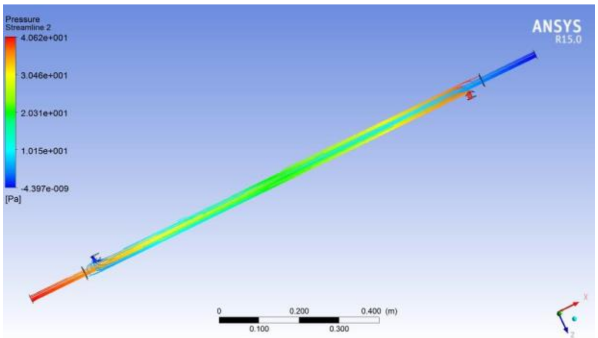


Fig 5.14 Temperature variation
Fig 5.15 Pressure variation

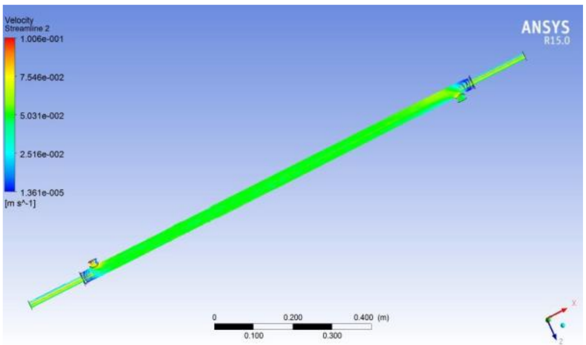


Fig 5.16 Velocity variation

5.4.2 Sample Calculation for Counter Flow:

Thi = 312 K Tci = 300 K
Tho =306.80013 K Tco = 303.991287 K
Mh = (1.2)/60 = 0.02 litres/s
Mc = (1)/60 = 0.016667 litres/s
Cph = 2813 J/kgk Cpc = 4174 J/kgk
Qh = Mh Cph (Thi –Tho) watts
Qh =292.5442361 Watts.
Qc = Mc Cpc (Tco– Tci) watts
Qc = 277.6605323 Watts
Qa = (Qh + Qc)/2= 285.102384 Watts
LMTD = (θ2 – θ1)/ Log (θ2/θ1)
θ2 = Tho – Tci =306.80013 – 300 = 6.800138 K
θ1 = Thi – Tco = 312 - 303.991287 = 8.0088 K
LMTD = (θ2 – θ1)/ Log (θ2/θ1)= 7.38795723 K
Overall heat transfer coefficient:
Uo = Qa/ (Ao* LMTD)
= (285.102384) / (0.0863= 447.1628144 W /m^2K

5.4.3 Counter flow study results:

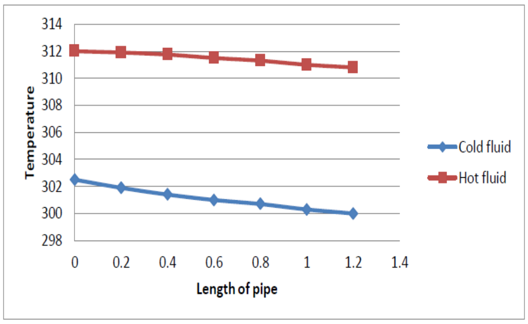
For counter flow heat exchangers the hotter fluid will lower in temperature as it loses heat to the cooler fluid which will then rise in temperature due to the heat transfer.

Figure 5.17 shows this gradual temperature change in both flow paths. This is the correct curve form already proven for counter-current flow heat exchangers.

In the figure 5.18 shows that heat transfer co-efficient is directly proportional to mass flow rate i.e. by increasing mass flow rate, the Overall heat transfer co-efficient increases. As shown in figure, Glycerin CFD acquired better heat transfer co-efficient compared to other organic fluids on increasing mass flow rate.

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

In the figure 5.19 shows that LMTD is directly proportional to mass flow rate i.e. by increasing mass flow rate, LMTD increases. As shown in figure, Glycerin CFD acquired better LMTD compared to other organic fluids on increasing mass flow rate.



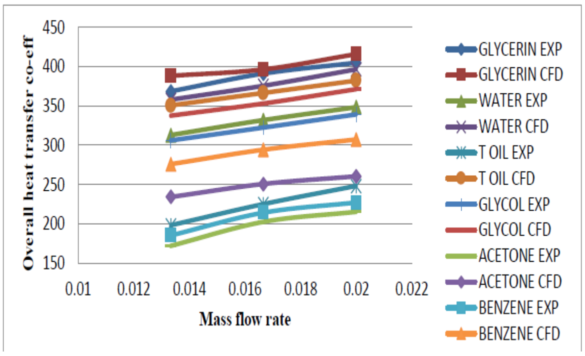


Fig 5.17 Counter Flow Profile Curve Fig5.18Overall heat transfer Co-efficient vs.Mass flow rate

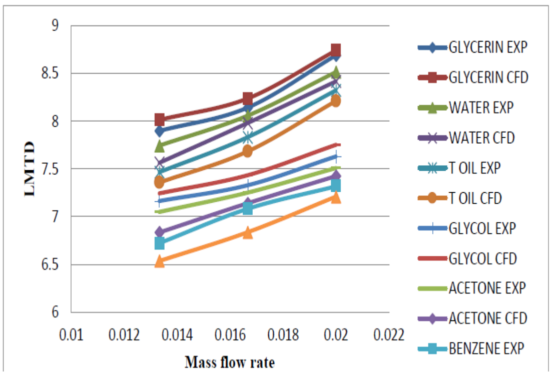


Fig 5.19 LMTD vs. Mass flow rate

5.4.4 Heat exchanger with baffle segments:

The baffle segmental is introduced above the outer surface of the inner pipe at different angular orientations like 0º, 45º, 90º and 135 º. The experiment is performed in parallel flow and counter flow arrangements for aluminum material with glycerin fluid.

As shown in figure 5.20, Glycerin in parallel flow acquired better LMTD compared to other organic fluids on increasing mass flow rate. From the figure that LMTD is directly proportional to mass flow rate i.e. by increasing mass flow rate, the LMTD increases

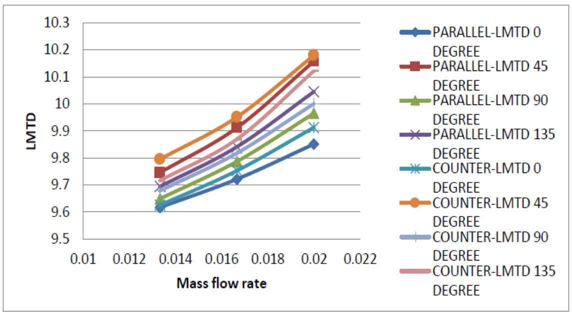


Fig.5.20 LMTD vs. Mass flow rate

6.CONCLUSION:

The following are conclusions derived using CFD analysis:

- It is concluded that Double pipe heat exchanger using Aluminium material with glycerine as fluid is more efficient.
- Using Aluminium tube material and In Parallel flow arrangement at a flow rate of 0.02 m/s by using Glycerine as a fluid -In Experiment Analysis, fluid acquired better rate of heat transfer is 252.88 watts.

-In CFD Analysis, fluid acquired better rate of heat transfer is 267.46 watts.

- Using Aluminium tube material and In Counter flow arrangement at a flow rate of 0.02 m/s by using Glycerine as a fluid
  -In Experiment Analysis, fluid acquired better rate of heat transfer is 234.20 watts.
  -In CFD Analysis, fluid acquired better rate of heat transfer is 285.10 watts.

- 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. At 45º Angular orientation fluid give better heat transfer.

- In Aluminium tube material inserted with Baffle segmental at an angle of 45º counter flow arrangement by using Glycerine fluid.
  -At a flow rate of 0.02 m/s fluid acquired better rate of heat transfer is 8494.884 watts.
  -At a flow rate of 2.4 m/s fluid acquired better rate of heat transfer is 11821.265 watts.

- The study results resolved that the double pipe heat exchanger with baffle segmental at 45º angular orientation using glycerine 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.