Journal of the Maharaja Sayajirao University of Baroda
ISSN: 0025-0422

HDPE DOUBLE PIPE HEAT EXCHANGER

Dr. D.V. Sreekanth[1], Dr. P. Santosh Kumar Patra[2]

[1]Professor, Mechanical Department, St. Martin's Engineering College, Secunderabad, Telangana
[2]Principal & Professor, CSE Department, St. Martin's Engineering College, Secunderabad, Telangana
E- Mail: dvsrikanth@smec.ac.in

Abstract

A heat exchanger is a device used to transfer energy from two or more fluids, from a solid surface and a fluid, or from solid particulates and a fluid, at distinctive temperatures and which are in thermal contact. Double pipe heat exchanger is the most common type of heat exchanger used in chemical Industries and especially where the heat transfer requirement is minimum. Generally, Plastics are not considered for heat applications as they are bad conductors for heat. But researches proved that though the Plastic is thermal insulator, they can transfer the heat efficiently if there is thermal resistance. In order to get the efficient heat transfer the heat exchanger is designed with the large surface area and thin wall thickness of the heat exchanger.

The aim of this project is to study the heat transfer of HDPE Heat exchanger by Computational Fluid Dynamics(CFD) Using ANSYS 18.1 software by plotting temperature, Velocity contours from CFD Analysis. CFD is the very useful technique to simulate the heat transfer of the heat exchangers. It will predict the heat transfer of the heat exchangers without building the prototype, hence it reduces the cost, time etc. The results obtained from the CFD analysis is validated using the NTU method to check the accuracy of the results.

Keywords: Double Pipe Heat Exchanger, Parallel Flow, Counter Flow, CFD, NTU

1. Introduction

A heat exchanger is a device used to transfer energy from two or more fluids, from a solid surface and a fluid, or from solid particulates and a fluid, at distinctive temperatures and which are in thermal contact. Heat exchangers are one of the important engineering devices in process industries since the efficiency and economy of the process largely depend on the performance of the heat exchangers.

1.1 Classification of Heat Exchangers:

The heat exchangers are classified into different types based on the nature of heat exchange process , direction of flow, based on the state of the fluids, based on the design of heat exchangers.

Based on the nature of heat exchange process:

a. Direct contact Heat exchanger
b. Regenerator
c. Recurperator

Based on the design of Heat exchangers are classified as follows:

1. Concentric Tubes or Double Pipe
2. Shell and Tube
3. Multiple shell and tube pass

Based on the flow direction Heat exchangers are classified into two types.

1. Parallel Flow: In this type of flow the direction of both the hot and cold fluids is in the same direction.
2. Counter Flow: In this type of flow the direction of both the hot and cold fluids is in the opposite direction.

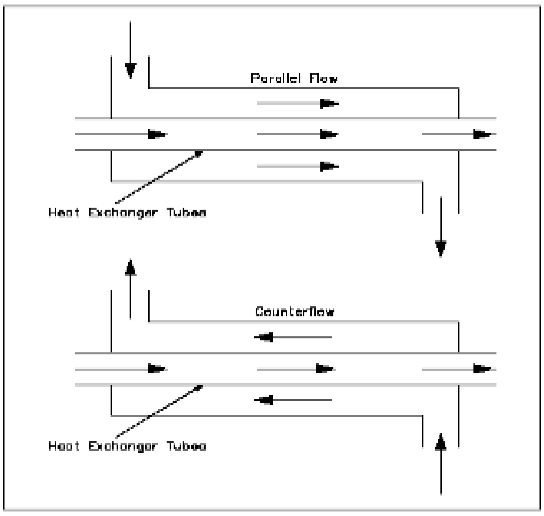

Fig1: Shows Parallel and Counter flow

Based on the physical state of fluids the heat exchangers are classified as follows:
a) condensers
b) Evaporators

1.2 **High Density Polyethylene:**

High Density Polyethylene is the mostly used Plastic polymer now-a-days. It is widely used to manufacture the products like plastic bottles etc. But the plastic heat exchangers made with HDPE have also started emerging. It is known that plastics are bad conductors for heat as they having very less thermal conductivity. The thermal conductivity of the material depends upon the surface area of the heat exchanger and the thickness of the walls of the heat exchanger. So by using the heat exchangers with large surface area and minimum thickness. Based on these parameters if we use the heat exchangers made with HDPE polymers it can be used to design heat exchangers with less weight and cost compare to its metal counter parts.

Advantages:

1. High Resistance to Corrosion
2. Resistance to Fouling
3. Light Weight and Less Cost
4. Non-reactive to many chemicals.

1.1 Design of Double Pipe Heat Exchanger

The 3D model of the Double pipe heat exchanger is designed in SolidWorks 2019 modelling software. The model is as shown in the Fig. 2.

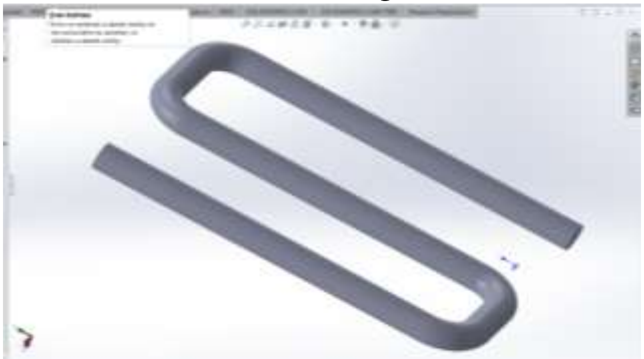

fig 2 3D model of Heat Exchanger

2.1 Design Specifications of Double Pipe Heat Exchanger:

Diameter of Outer Pipe: $D_{IO=}$ 32 mm
Diameter of Inner Pipe: D_{II}=25.4 mm
Thickness of Wall : t = 1 mm

3. Computational fluid dynamics (CFD): Computational fluid dynamics (CFD) is the science of predicting the fluid flow, heat and mass transfer, chemical reactions etc. using the equations conservation of mass, energy momentum. These equations are used to describe how the velocity, pressure, temperature, and density of a moving fluid are related. CFD can be used in all stages of product development especially in the development of the heat exchangers. It reduces the cost and time for testing with practical experiments.

3.1 MESHING

Meshing is defined as the process of dividing the given structure to finite elements. It is performed in ANSYS Mechanical to discretize the elements of any given model. It is very important to obtain good quality of mesh to enhance good results. Using different set of parameters, the relative finer mesh is obtained on the model of Double Pipe heat exchanger. The 3D type of meshing is used for this model. The mesh size of 4mm is taken for the accurate results. 169388 no. of nodes and 142345 no. of elements are generated. The boundary conditions for the inlet and outlet of the fluid is fixed in the ANSYS Mechanical. The utmost care is taken to obtain the good quality of mesh as it affects the results in the solver. The meshing methods like inflation, body sizing is used for obtaining better quality of mesh. The inlet and outlet portions of the heat exchanger is fixed in the ANSYS Mechanical.

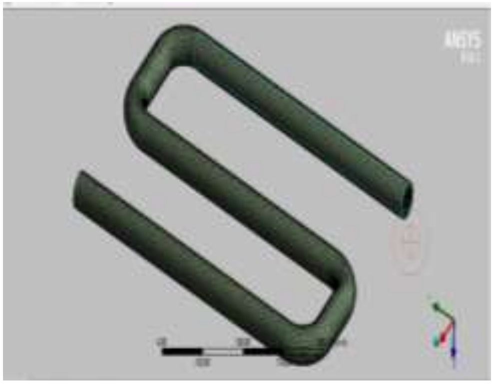

Fig 3: Meshing model of fluid flow

3.2 ANSYS FLUENT

Fluent software consists all the features needed to model the flow, heat transfer and for other industrial applications. In the Fluent software we need to define the properties of the material, appropriate physical models, boundary conditions, solver and convergence controls. The discretised conservation equations are solved until the convergence is reached. The accuracy of the convergence depends upon the quality of the mesh, accuracy of the physical model etc.

The following parameters set in the ANSYS FLUENT before starting solving the solution.

3.2.1 **Material**:

The materials for doing the CFD analysis are chosen from the Fluent database available in Ansys Fluent.

HDPE: It is used for the design of Double Pipe Heat exchanger. The properties of HDPE are as follows:

Thermal Conductivity - 0.5 W/m-k

Specific Heat (Cp) - 1900 J/KG-K

Density - 8978 Kg/m3

Water: Water is used as both the hot and cold fluid in the double pipe heat exchanger. The properties of the water are as follows

Thermal Conductivity - 91 W/m-k

SPECIFIC HEAT (Cp) - 460 KJ/K

Density - 8900 Kg/m3

3.2.2 BOUNDARY CONDITIONS

The boundary conditions are assumed considering the material properties of both the hot and cold fluid. for both the parallel flow and Counter flow are considered as follows:

Mass flow rate of hot fluid – 1.3 kg/s

Mass flow rate of cold fluid– 0.2 kg/s

Inlet Temperature of Hot Fluid =

$T_{h,i}$ = 343 K

Inlet temperature of cold fluid=

$T_{c,i}$ =293K

Heat flux around the surface - 0 w/m^2

4. Result:

The results are obtained in the Ansys fluent software after few iterations. The temperature contours are plotted in the CFD Post Solver. In the temperature contours shown in fig4 and fig 5, the maximum temperatures are indicated by the red colour and the minimum temperatures are indicated in blue colour. The heat transfer of the fluids is shown in these temperature plots and the change in colour represents the heat transfer taking place between the fluids.

The following are the results obtained in parallel and counter flow cases.

4.1 Parallel Flow:

The solution is converged after the 200 iterations in the Ansys Fluent Software. The temperature contours are plotted in the mid plane of Z – axis which are shown in fig. In this case the cold water is flown in the annulus of the pipe and the hot water is flown inside the inner pipe of the Double Pipe Heat Exchanger along the same direction. The following are the results of the

Outlet temperature of hot fluid - $T_{h,o}$ =325K

Outlet temperature of cold fluid - $T_{c,0}$= 296 K

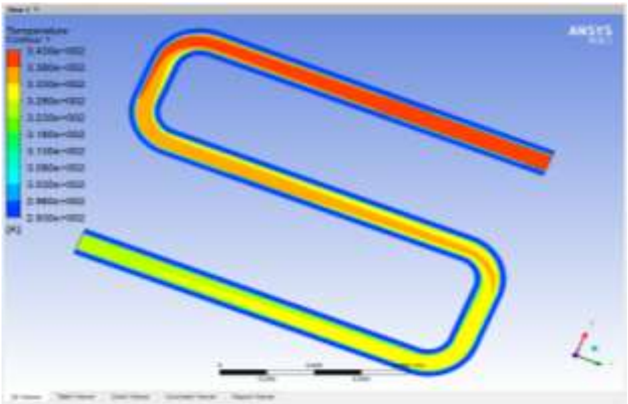

Fig 4. Temperature Contour for Parallel Flow

Net	0.024267046
Average of Facet Values Static Temperature	(k)
inlet1_fluid_flow__fluent__	293
inlet2_fluid_flow__fluent__	343
outlet1_fluid_flow__fluent__	295.63148
outlet2_fluid_flow__fluent__	325.74533
Net	300.55838

Fig 5. Facet values of temperatures

4.2 Counter Flow:

The solution is converged after the 200 iterations in the Ansys Fluent Software. The temperature contours are plotted in the mid plane of Z – axis which are shown in fig. In this case the cold water is flown in the annulus of the pipe and the hot water is flown inside the inner pipe of the Double Pipe Heat Exchanger along the opposite direction to each. The following are the results obtained from the Fluent Solver

Outlet temperature of hot fluid - $T_{h,o}$ =326 K

Outlet temperature of cold fluid - $T_{c,0}$= 295 K

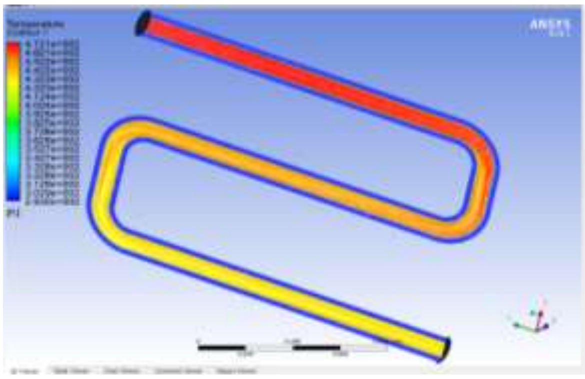

Fig 6. Temperature Contour for Counter Flow

Average of Facet Values Static Temperature	(k)
inlet1_fluid_flow__fluent__	293
inlet2_fluid_flow__fluent__	343
outlet1_fluid_flow__fluent__	295.64491
outlet2_fluid_flow__fluent__	325.62342

Fig 7. Facet values of temperatures

5. Result Validation:

The results obtained from the Ansys fluent is validated by comparing with the theoretical calculations to ensure the accuracy of the solution obtained from the ANSYS Fluent Solver.

The NTU METHOD is used to validate the results obtained from the ANSYS Fluent .

The following equations are similar to both the parallel and counter flows:

Heat transfer rate is given by:

$Q1 = m_c \times CP_c \times (T_{c,o} - T_{c,i}))$…………(1)

$Q2 = m_h \times CP_h \times (T_{hot\ In} - T_{hot\ out})$ ….(2)

We know that heat lost by hot fluid= heat gained by cold fluid. Therefore by equating the equations (1) and (2),we get

$$Q1 = Q2 \quad \text{………} \quad (3)$$

$$m_c \times CP_c \times (T_{c,o} - T_{c,i})\text{………..}(4)$$

$$m_h \times CP_h \times (T_{h,i} - T_{h,o})\text{…….}(5)$$

5.1 NTU METHOD:

NTU stands for Number of Transfer of Units. This method is very efficient to use to design the heat exchangers when the final temperatures of the hot and cold fluids are not known. We can also find the outlet temperatures and heat flow rates with the help of some charts but it would be time taking as it involves large number of trials and it also least accurate. In this case the Effectiveness –NTU method is used.

The effectiveness of a heat exchanger is defined as the ratio of actual heat transfer to the maximum possible heat transfer. It is expressed as in terms of expression as follows:

$$\epsilon = \frac{q}{q_{\max}} \text{…………..}(6)$$

The actual heat transfer is obtained from the equation. The maximum heat transfer rate is the rate of heat transfer that a counter flow heat exchanger of infinite area would transfer with given inlet temperature, specific heats and flow rates. It is considered that maximum heat transfer would be possible if one of the fluids undergoes the temperature change equal to the maximum temperature difference present in the heat exchanger.

The heat capacity rates i.e. mass flow rate multiplied by specific heats are expressed as C_h and C_c for the hot and cold fluids respectively and smaller heat capacity rate I expressed as $C_{\min}$

$$C_{\min} = \min[\dot{m}_c c_{p,c}, \dot{m}_h c_{p,h}] \text{…………}(7)$$

Therefore, the maximum heat transfer that would possible between the two fluids per unit time is given by

$$q_{\max} = C_{\min}(T_{h,i} - T_{c,i}) \text{…………}(8)$$

$C_{\min}$ is the lowest heat capacity rate of the fluid that would undergo maximum temperature change. The other fluid would undergo temperature change throughout the length of the heat exchanger .

The actual heat transfer is expressed as follows:

$$q = C_h(T_{h,i} - T_{h,o}) = C_c(T_{c,o} - T_{c,i}) \text{….}(9)$$

The effectiveness of a heat exchanger is a dimensionless quantity which always lies between 0 and 1. From this we can calculate the heat transferred between the fluids if we know the effectiveness of the heat exchanger and inlet temperatures of the fluid by the following equation

$$q = \epsilon C_{\min}(T_{h,i} - T_{c,i}) \text{…………}(10)$$

The effectiveness of the heat exchanger can be expressed in the form of specific heat rates as follows:)

$$C_r = \frac{C_{\min}}{C_{\max}} \text{…………….}(11)$$

The Number of Transfer Units (NTU) is given by

$$NTU = \frac{UA}{C_{\min}} \quad \text{............ (12)}$$

Where,

U is the overall heat transfer coefficient and A is the heat transfer area of the given heat exchanger.

The effectiveness of a parallel flow heat exchanger in terms of the NTU is given as follows:

$$\epsilon = \frac{1 - \exp[-NTU(1 + C_r)]}{1 + C_r} \quad \text{.......(13)}$$

The effectiveness of a counter flow heat exchanger in terms of the NTU is given as follows:

$$\epsilon = \frac{1 - \exp[-NTU(1 - C_r)]}{1 - C_r \exp[-NTU(1 - C_r)]} \quad \text{...(14)}$$

NTU is the dimensionless parameter. It is the measure of the heat transfer size of the heat exchanger. The value of the NTU is directly proportional to the thermodynamic limit of operation of the heat exchanger. From the equations we will get the theoretical values of outlet temperatures of the hot and cold fluid for both the parallel and counter flow. Based on the temperatures obtained theoretical heat transfer of the heat exchanger is calculated

5.2 VALIDATION TABLE:

5.2.1 For Parallel Flow:

S. NO	PARAMETER	EXPERIMENT	THEORETICAL
1	$T_{h,o}$	325 K	333 K
2	$T_{c,o}$	296 K	305 K
3	HEAT TRANSFER	112.78 KJ	93.99 KJ

5.2.2 For Counter Flow:

S. NO	PARAMETER	EXPERIMANTAL	THEORETICAL
1	$T_{h,o}$	326 K	333 K
2	$T_{c,o}$	295 K	305 K
3	HEAT TRANSFER	81.12 KJ	62.6 KJ

6. CONCLUSION:

The main objective of this project is to study the performance of the HDPE Double pipe heat exchanger by using CFD. Here we obtained the final results of the performance of the Heat Exchanger. The heat transfer of the heat exchanger is calculated and is compared with the theoretical results and found satisfactory as the values are almost equal. We can conclude that HDPE exhibits good heat transfer characterstics for the given specifiacations of the heat exchanger and with the certain assumptions.

7. REFERENCES

[1] Nitin S gokhale ,Sanjay S deshpande - practical finite element analysis

[2] Numerical Problems related to heat exchanger performance

[3] Kuruva Umamahesh, Design And Anaysis Of Double Pipe Heat Exchanger Using Computational Method.

[4] R. K. Shah, D. P. Sekulic, Fundamentals of Heat Exchanger Design, Wiley, 2003

[5] P. Wais, J. Taler, Fin shape optimization in tube heat exchangers by means of CFD program, 2nd International Conference on Engineering Optimization, Sept. 6 - 9, Lisbon, Portugal, (2010) 1-10.