International Journal of Sciences: Basic and Applied Research (IJSBAR)

ISSN 2307-4531
(Print & Online)

http://gssrr.org/index.php?journal=JournalOfBasicAndApplied

Computational Fluid Dynamics of Parallel Flow Heat Exchanger

Kranthi Kumar Mamidala [a]*, Srikesh [b]

[a] UG student at GITAM University, Hyderabad Campus, Rudraram, 502102, India

[b] UG student at GITAM University, Hyderabad Campus, Rudraram, 502102, India

[a] Email: kranthi.kmamidala@gmail.com

[b] Email: msrikesh11@gmail.com

Abstract

Heat exchangers are a class of Mechanical Components where heat transfer process takes place between cold and hot objects. As the machines need to be cooled heat is extracted. But this heat can be utilised in many ways in industries. So the best component available to extract heat from the exhaust gasses is by the use of Heat Exchangers. If the efficiency of heat exchanger is good maximum heat utilisation from waste gasses is achieved. The simplest heat exchanger available is parallel and counter flow indirect contact type heat exchange. But until now, there has been no attempt made in studying the heat pattern in a simple heat exchanger. So our project aims in analysis, of heat flow pattern of heat exchanger with the help of Fluent Software. We also proceed to determine the percentage error in experimental and analytical value obtained. In order to validate our results from the Fluent Analysis we have directly carried out the experiment on the parallel flow heat exchanger set up.

Keywords: Thermal Engineering.

1. Introduction

Heat exchanger is a Mechanical component wherein heat transfer takes place between two liquids. By the laws of heat transfer it is very much known that heat from a hot fluid is transferred naturally to cold fluid. This process of heat exchange takes place until an equilibrium state is attained by the two heat transfer liquids [1].

----

* Corresponding author

There are many ways of classifying heat exchanger. But on a broad general classification heat exchangers are of 2 types.

1) Direct type Heat Exchanger
2) In direct type Heat Exchanger

Direct type of heat exchanger is where heat exchange process takes place between 2 fluids because of their physical contact.

There is no physical contact between two heat exchanging liquids in indirect type of heat exchanger. Here heat exchange process takes place through an intermediate substance. These heat exchangers are again classified into

1) Regenerative type of Heat Exchanger
2) Recuperative type of Heat Exchanger

In regenerative type hot fluid and cold fluid flows into heat exchanger one after another. But in recuperative type both the fluids flow simultaneously [1].

1.1 Need for analysis of heat exchanger by CFD

Heat is considered to be a form of energy. So as the goes energy is neither nor destroyed, we should not allow the heat to go out as waste. One of the way to conserve the heat energy which is going out as waste in the exhaust gasses is to extract the heat and transfer it to a cold body. One of the important components of heat transfer system is Heat Exchanger. Heat exchanger is widely used equipment in various industries such as process, power generation, petroleum refining, chemicals and paper. According to a market survey conducted in Europe, it accounts for about 42% of the market share. It also used for the heat exchange process in Nuclear industry as well. Hence there arises a need for analysis of Heat Exchanger [1].

2. Computational Fluid Dynamics

Computational Fluid Dynamics CFD is one of the best suited software for the analysis of flow patterns which involve the dynamic parameters in the flow. It is very stable robust and accurate in providing the required outputs. Generally experiments on moving fluid particles are not feasible. Dynamics study on moving particles involves many complex calculations and large variables. But their analysis can be easily carried with the help of CFD[6].

2.1 Working of CFD

CFD is based on Finite Volume Method. The area in the flow pattern for the analysis is first modelled in any of the modelling software. In terms of CFD this is called as Computational domain. This computational domain is discretized into many small elements of finite volume. This process of dividing the domain is termed as meshing.

Hence here each volume is considered to be a control volume. The control volume have certain properties like mass, momentum, energy, turbulence quantities, and mixture fractions, species concentrations and material properties. Based on the flow problem we pick up the control volume properties to be analysed and solve the flow problem. The theoretical flow in control volume is represented physically with the help of numerically solvable partial differential equations. These equations govern the flow of fluid in the computational domain. These equations also undergo the discretization process for the flow analysis.

There are three equations which are applicable commonly to all fluid dynamics problems are the conservation of mass, momentum and the energy equations. Equations when represented in differential form [6].

Continuity equation: $$\frac{\partial \rho}{\partial t} + \nabla (\rho\, u) = 0 \quad (1)$$

Momentum equation $$\rho(\frac{dv}{dt} + v.\nabla v) = - \nabla p + \nabla T + f \quad (2)$$

Energy equation: $$\frac{\partial}{\partial t}(\rho h) + \frac{\partial}{\partial x_j}(\rho u_j h) = - \frac{\partial q_j}{\partial x_j} + \frac{\partial \rho}{\partial t} + u_j\frac{\partial P}{\partial x_j} + \tau_{ij}\frac{\partial u_i}{\partial x_{ij}} \quad [6] \quad (3)$$

Above equations along with many other equations depending on the type and problem of flow are considered as PDE’s Partial Differential Equations. These equations when subjected to discretization give out a set of algebraic equations. These algebraic equations contain terms that have been used for specifications of that particular flow and also these equations contain our required output parameters. After solving these equations, the solution of these algebraic equations is found with the CFD program called as FLUENT. This process of solving the flow problem involves generation of output at each element nodes for accurate and precise results.

Analysis in CFD involves repeated and sequential solving of algebraic equations. The main purpose for this repeated iterations is to improve the quality of solution constantly after each and every iterations. The process of iterations is continued until convincing values of the global residuals are obtained. Global residuals are difference between the values of output parameters obtained in the current and previous solution which are averaged over the entire computational domain. After continuous iterations a state of “convergence” is obtained where the residuals have decrease by 4-5 orders of magnitude. One of the most important points to be considered when working with CFD software is that the quality of its output is completely dependent on the quality of its input. Hence when best quality inputs are given to the solver best quality outputs are obtained for the analysis [6].

2.2 K-ξ turbulence model

This model is most reliable and robust very frequently used CFD platform. This model of analysis has some special from others models like K- omega, RANS, Large Eddy simulations etc. In the K- ξ model K represents the kinetic energy and ξ represents the turbulence. Both of the parameters are determined with the help of certain following equations [7].

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j}\{[\mu+\frac{\mu_t}{\sigma_k}]\frac{\partial k}{\partial x_j}\} + G_k + G_b - \rho\varepsilon - Y_m + S_k \quad (4)$$

$$\frac{\partial}{\partial t}(\rho\varepsilon) + \frac{\partial}{\partial x_i}(\rho\varepsilon u_i) = \frac{\partial}{\partial x_j}\{[\mu+\frac{\mu_t}{\sigma_\varepsilon}]\frac{\partial \varepsilon}{\partial x_j}\} + C_1\varepsilon\frac{\varepsilon}{K}(G_k+C_3\varepsilon G_b) - C_2\varepsilon\rho\frac{\epsilon 2}{K} + S_t \quad (5)$$

$$\mu_t = \rho C\mu\frac{K2}{\varepsilon} \quad (6)$$

$$G_k = -\rho u_i' u_j'\frac{\partial u_j}{\partial x_i} \quad (7)$$

$$S = \sqrt{2}S_{ij}S_{ij} \quad [7] \quad (8)$$

$$Gb = \beta gi\frac{\mu t}{Prt}\frac{\partial T}{\partial xi} \quad (9)$$

$$Y_m = 2\,\rho\epsilon M_t^2 \quad (10)$$

NOMENCLATURE:

ρ density

u initial velocity

p pressure

f body force

Q vector variable

F vector fluxes

V volume of control volume

A area of control volume

R_i equation residual

W_i weight vector

V^e volume of element.

3. Description of Heat exchanger experimental setup

Process of heat exchanger experiment is carried out in Heat and Mass Transfer laboratory. Whole heat exchanger set up is placed on a firm stand. Dimensions of the stand are 52 by 70 inches.

Height of the base is 28 inches with the height of the experimental set up as 20 inches. Water supply needed for the experiment is obtained from an over head tank available outside the laboratory. Water supply is obtained through connecting PVC pipes of 2 inch diameter. There is a regulating valve with which water flow is controlled [2].

Heat Exchanger Experiment set up has a small diameter tube which is fixed inside another bigger diameter tube concentrically. The outer tube is insulated to reduce the heat losses from its surface to the surroundings.

Working fluids in the experiment are Hot fluid and Cold Fluid. Heat is transferred from Hot fluid to Cold Fluid. The cold water is supplied connecting the inlet to the common tap supply and its flow is controlled by providing inlet and outlet values which are placed at the ends. For the supply hot water, a small water tank is provided with heater immersed in it.

The water level in the hot water tank and its temperature is maintained constant as required by controlling the power supply and a valve through which cold water tank. The valves arrangement is made in such a way, that the flow can be made parallel or counter as desired in the same experimental set up. The direction of hot water flow through the inner pipe is maintained same and the direction of cold water flow is made parallel or counter as desired by changing the valve positions. The two thermometers are provided to measure the inlet and outlet [2].

temperatures of hot water (T_{hi} and T_{ho}) and other two measure the inlet and outlet temperatures of cold water (T_{ci} and T_{co}). An anemometer is also provided to control the mass flow rate of water through the pipe. This has to be adjusted to change the mass flow rate while carrying the experiment [2].

3.1 Geometrical specifications of heat exchanger

The inner tube through which the hot water is flowing is made of Copper material. Outer tube through which cold water is flowing is made of Stainless Steel.

Table 1: specifications of Heat exchanger

Components	Dimension
Hot Water tube (Copper)	Length : 1100 mm Inner Diameter : 21mm Outer Diameter: 25mm
Thickness	4mm
Cold Water Tube (Stainless Steel)	Length : 1100mm Inner diameter : 32mm Outer Diameter : 38mm
Thickness	6mm

Table 2: experimental values

Sl. No.	**water flow rate in LPM (hot)**	**water flow rate in LPM (cold)**	**T_{CI}°C**	**T_{CO}°C**	**T_{HI}°C**	**T_{HO}°C**
1	0.4	0.2	28	38.5	59.75	45.5
2	0.4	0.4	28	35.1	58.5	46.2
3	0.4	0.6	27.9	33.5	58.4	47.8

3.2 CFD Analysis

First the computational domain is created in any modelling software like CATIA, PRO-E, GAMBIT etc.. Here we have chosen GAMBIT 2.3.16. Now this domain later extracted to Fluent 6.3.26 software for analysis. Here is the 3D model of Heat exchanger created using GAMBIT software.

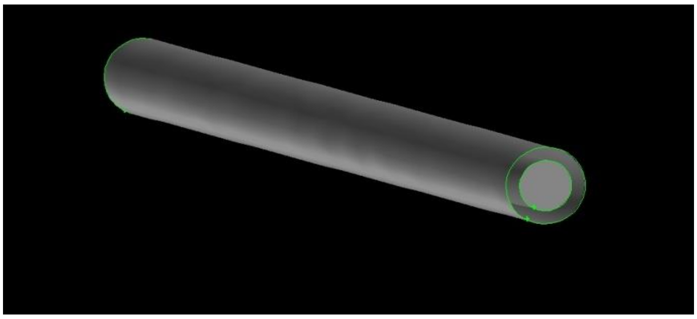

Figure 1: 3D model of Heat exchanger

Now this model is extracted on to fluent. All the required specifications and parameters are incorporated into it. And the analysis is iterated for 1000 values.

3.3 Formulae used

i. $Q_h = m_h C_{ph} (T_{hi} - T_{ho})$ in watts.

Where, m_h= mass flow rate of hot water in LPs.

C_{ph}= Specific heat of water = 4187 J/kg ^oK

T_{hi}= Inlet temperature of hot water

T_{ho}= Outlet temperature of hot water

ii. $Q_c = m_c C_{pc} (T_{co} - T_{ci})$ in watts.

Where, m_c = mass flow rate of hot water in LPs.

C_{pc}= Specific heat of water = 4187 J/kg ^oK

T_{ci}= Inlet temperature of hot water

T_{co}= Outlet temperature of hot water iii.Q= $(Q_h + Q_c) / 2$ Watts

iv. LMTD $= \frac{\theta 2 - \theta 1}{Ln\left(\frac{\theta 2}{\theta 1}\right)}$

v. Uo = Q / (Ao x LMTD) W/m^2 k [3]

4. Result

As mentioned earlier that we have analysed the heat exchanger for different mass flow rates. In our project we have assumed the flow rate of hot water as constant and the flow rate of cold water is varied.

Case 1: hot water flow rate = 0.4 lpm cold water flow rate = 0.2 lpm

Case 2: hot water flow rate = 0.4 lpm cold water flow rate = 0.4 lpm

Case 2: hot water flow rate = 0.4 lpm cold water flow rate = 0.6 lpm

Analysis about the heat exchanger is carried out by analysing the graphs which are plotted. Values in the graph have been determined by the analysis results. We have considered the analysis of cold water, it because mass flow rate of cold water is changed in every case. We have also considered the parameter of graphical analysis at various lengths of the heat exchanger tube. This step has been taken to find out the overall distribution of parameter along the length of heat exchanger tube. All parameters tabular column is given below

Table 3: values during experiment

Mh(lps)	Mc(lps)	Tci	Tco	Thi	Tho	LMTD	Uo
0.0066	0.0033	301	311	330.5	318.5	36.98	73.53
		300	312	331.6	319.7	38.97	73.51
		302	310.6	333.2	319.3	40.56	71.82
		301	312.4	335	320.5	41.57	77.77
0.00667	0.00667	300.9	308.8	326.7	321.1	41.96	51.50
		301	308.2	329.7	320.5	44.56	58.91
		300.8	307.6	333.4	320.1	48.28	54.64
		301	307.8	336.2	319.4	48.95	77.17
0.00667	0.01	300.9	307.6	336.1	319.8	49.91	84.86
		301	306.4	333.5	319.4	49.00	72.79
		300.8	306.7	330.7	320.1	47.33	66.09
		301	306.5	325.7	320	42.68	52.62

4.1 Temperature Analysis

Case1

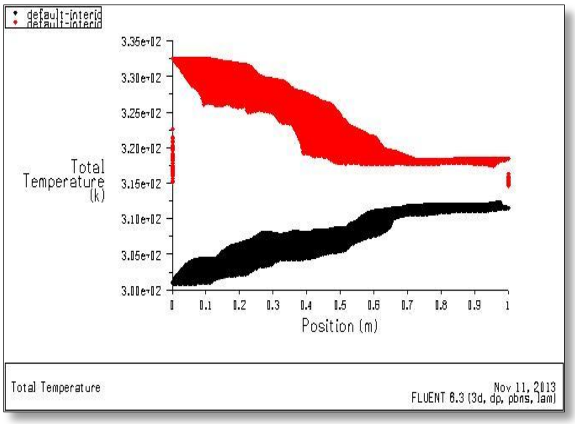

Figure 2: Temperature Distribution

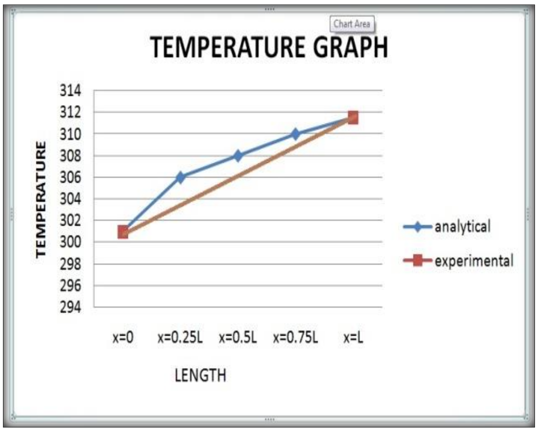

Figure 3: Temperature Graph

Case 2

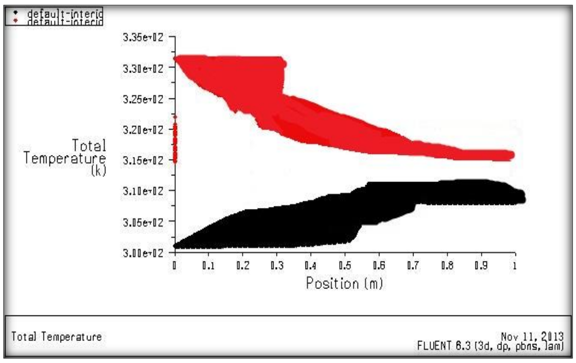

Figure 4: Temperature Distribution

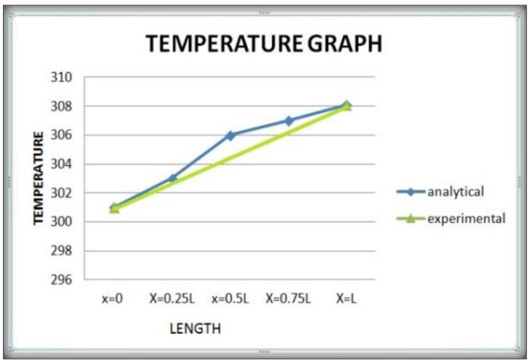

Figure 5: Temperature Graph

Case 3

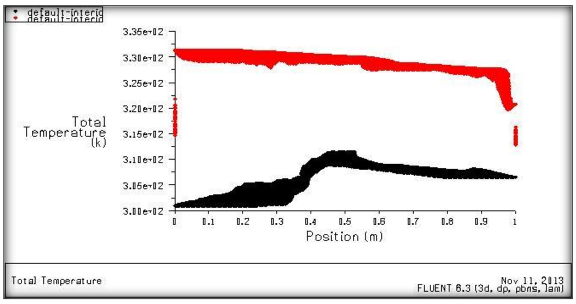

Figure 6: Temperature Distribution

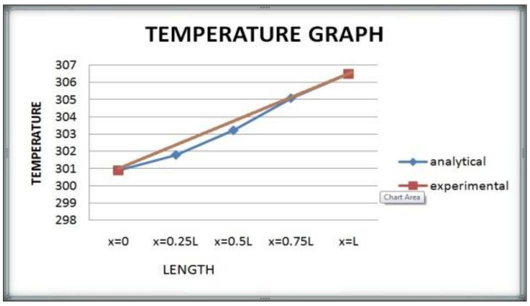

Figure 7: Temperature Graph

4.2 Heat transfer coefficient Analysis

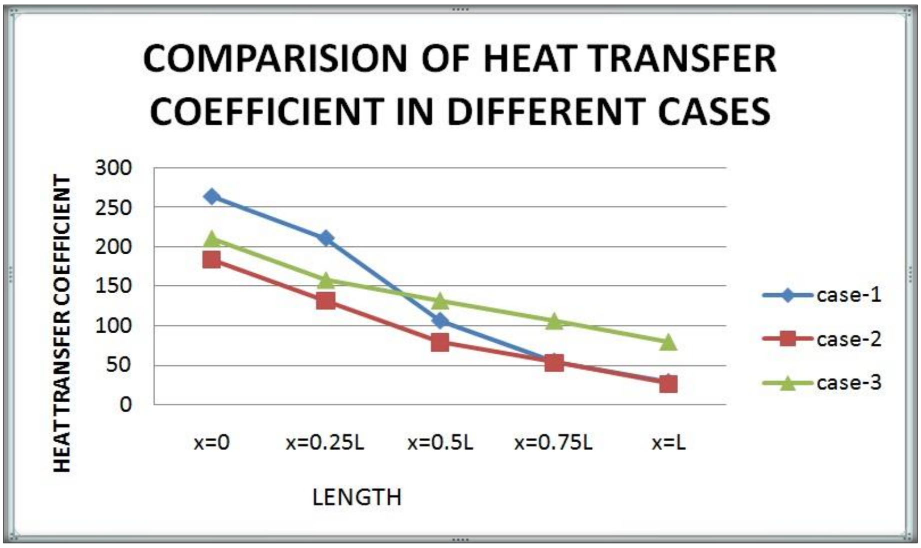

Figure 2: Comparison of Heat transfer coefficient in different cases

5. Conclusion

All the above sections indicate various plots and graphs which shows the analysis of heat exchanger. The parameter which is varied for all the 3 cases is the mass flow rate of cold water. We are aware that when a water stream is allowed to flow on a surface boundary layer phenomenon formation takes place. Parameters which are analysed in our project are dependent on boundary layer formation. Surface heat flux and heat transfer co-efficient depended on the thickness of boundary layer formed. When the stream of water is flowing in a pipe boundary layer is formed from both sides of diameter. At certain length of the pipe both the boundary layers merge and from there it is called as fully developed region. So in this region velocity is not dependent on radial direction but only dependent in axial direction.

Heat transfer co-efficient is inversely proportional to the thickness of boundary layer, so as the velocity of cold water increases, thickness of boundary layer decreases and the length at which the fully developed region increases. Hence the length at which the boundary layer merges becomes shorter from case 1 to case 3. Hence there is increment in heat transfer co-efficient and surface heat flux value from case 1 to case 3.

As for the temperature analysis is considered, the shape of the temperature plot is similar to that of general temperature plot of parallel flow heat exchanger. but there has been small 0.9% error from the analytical and experimental values.

6. Future Scope of Work

Previous sections have given various parameters of heat exchangers. This has deal with minimal part of heat exchanger analysis.

The efficiency of heat exchanger can be still more enhanced by many methods. Rate of heat transfer is dependent on area of heat exchanger also. So by increasing area of heat transfer or providing external area of contact heat exchanging process can be increased. We call the extended heat surfaces as FINS. Even as per the concepts various shapes of fins can also be experimented to provide best results. Research has to be carried out to determine best type of fin for improved heat transfer rate.

We have used cooper as the material for flow of hot water. So based on the thermal properties like thermal conductivity, co-efficient of heat transfer of copper heat transfer between hot and cold water is determined. But by replacing copper with a material which has better thermal characteristics we can also expedite the process of heat transfer. More analysis has to be worked out to get best suitable material for good heat transfer. This research may also involve fabrication of any new material with good thermal properties if we don’t find any better material than copper.

Reference

[1] Swapnanel sharma and D H Das. *CFD analysis of shell and tube type heat exchanger using triangular fins.* NIT Silcher:2012.

[2] Vikas kumar, D. Gangacharyulu, Parlapalli ms rao, R.S. Barve. *CFD analysis of cross flow air to air tube type heat exchanger.*Pune:2002

[3] E. Kruger, *CFD Analysis and experimental work on heat exchanger,* south africa:2001

[4] Ahmed F.Khudheyer and Mahmoud Sh. Mahmoud, *A numerical analysis of fin-tube plate heat exchanger by using CF.* Baghdad, Iraq:2011

[5] Khairun Hasmadi, *CFD Simulation of heat transfer in shell and tube heat exchanger,* Malaysia:2009

[6] website: https://en.wikipedia.org/wiki/Computational_fluid_dynamics, date accessed 9\2015.

[7] website: https://en.wikipedia.org/wiki/K-epsilon_turbulence_model, date accessed 9\2015.