RESEARCH ARTICLE OPEN ACCESS

Cfd Analysis on Concentric Tube Heat Exchanger in Parallel and Counter Flow Direction

Sneha.H.Dhoria, E. Manoj Kumar[2],I.V.S.Yeswanth[3], Lakshmi Jayanti[4]

Assistant Professor[1], Mechanical Engineering,RVR and JC College of Engineering, Guntur, India Associate Professor[2], Mechanical Engineering,Vardhaman Engineering College, Hyderabad, India Assistant Professor[3], Mechanical Engineering, QIS Institute of Technology, Ongole, India Assistant Professor[4], Chemical Engineering, RVR and JC College of Engineering, Guntur, India Corresponding author: Miss Sneha.H.Dhoria
Corresponding Auther: Sneha.H.Dhoria

ABSTRACT: A Heat Exchanger is a device which is used to transfer heat from one fluid to another, whether the fluids are separated by a solid wall so that they never mix, or the fluids are directly in contact. Every year Heat exchanger technology is growing to develop efficient, compact and economical heat exchangers, all over the world. Updating the community for this development needs an interaction. These days concentric tube heat exchangers are used with forced convection for lowering the working fluid's temperature by raising the cooling medium's temperature.The purpose of this project is to use ANSYS FLUENT 17.1 software and theoretical calculations to analyse the temperature drops as a function of both inlet velocity and inlet temperature and how each varies with the other. Each heat exchanger model was designed and simulated for both parallel flow and counter flow heat exchanger models. The results were compared between parallel and counter flow heat exchangers. CFD analysis was utilized to find the outlet temperatures of parallel and counter flow heat exchangers for the inlet velocity and inlet temperature of the fluid medium used. "Computational Fluid Dynamics (CFD) is a science of predicting fluid flow, heat transfer, mass transfer, and related phenomena by solving the mathematical equations which govern these processes using a numerical processes". These outlet temperature values obtained were used to determine the overall heat transfer co-efficient. Theoretical calculations are done by the values obtained through the experiment conducted on the heat exchanger setup for both parallel and counter flow.
Key words: Heat Exchangers, Parallel flow, Counter flow, Concentric Tube, CFD Analysis

---

Date of Submission: 01-06-2018 Date of acceptance:16-06-2018

I. INTRODUCTION

Heat Exchangers are used to transfer thermal energy form a high temperature fluid to low temperature fluid moving through a device. The temperature of the fluid changes as it passes through the heat exchanging device and hence the temperature of the dividing wall between the fluid also changes along the length of heat exchanger Examples:

- Condensers and boilers in steam plant
- Condensers and evaporators in refrigeration units
- Regenerators
- Automobile radiators
- Oil coolers of heat engine
- Milk chiller of a pasteurizing plant
- Several other industrial processes

Direction of Flow: According to the relative direction of two fluid streams the heat exchangers are classified into the following three categories:
1. Parallel flow
2. Counter flow
3. Cross – flow

1. Parallel flow heat exchangers:

In parallel flow heat exchangers the fluids both hot and cold travel in same direction. The flow arrangement for hot and cold fluids from inlet to outlet is shown in fig 1.1. In parallel flow heat exchangers the temperature difference from hot to clod fluid decreases. This type of heat exchangers requires large space and hence it is rarely used in practical applications. Eg: Oil coolers, oil heaters, water heaters etc, are examples of parallel flow heat exchanger.

2. Counter flow heat exchangers:

In a counter flow heat exchanger, the two hot and cold fluids enter at opposite ends. Theflow arrangement and temperature distribution for such a heat exchanger are shown schematically in fig. 1.2. the temperature difference between the two fluids remains more or less nearly constant. This type of heat exchanger, due to counter flow, gives maximum rate of heat transfer f or a given surface area. Hence such a heat exchangers are most favored for heating and cooling of fluids.

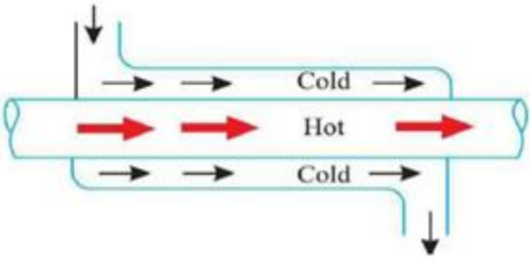


Fig. 1.2. Counter flow heat exchanger

3. Cross - flow heat exchanger:

When two fluids crosses one another in space at right angles such type of heat exchanger is known as cross flow heat exchanger. In cross flow heat exchanger there is no mixing of fluid streams and hot fluid flows in spate tubes and cold fluid is mixes perfectly as it flows through the exchanger. The temperature of this mixed fluid will be uniform across any section and will vary only in the direction of flow. The cooling unit of refrigeration system is an example of cross-flow heat exchanger

D.Bhanuchandrarao et al, [1] investigated CFD analysis and performance of parallel and counter flow in concentric tube heat exchangers.The results were compared between eachmodel and between parallel and counter flow with fouled piping. Turbulent flow was alsoanalysed during the development of the heat exchangers to determine its effect on heat transfer. While as expected the fouled heat exchanger had a lower performance and therefore cooled the working fluid less, the performance of the counter heat exchanger unexpectedly of the parallel heat exchanger.

Nice Thomachan et al, [2] investigated CFD analysis of tube in tube heat exchanger with fins. The heat transfer enhancement in a heat exchanger tube by installing fins on the outersurface of hot water tube. Design process for heat exchanger and insert has been carried out in AUTODESK INVENTOR, fluid domain is formed in ANSYS workbench, followed by meshing in default mesh tool of ANSYS. Boundary conditions were defining with appropriate material property in fluent software. After finding the solution the results are compared between the two designs for counter flow. According to results, it concluded that in case of fin is used, effectiveness alsoincreases. The reason behind maximum effectiveness was that due to use of fins, turbulence was increased as they allow more mixing of fluid layers and resulted in increase of heat transfer through the heat exchanger tube.

C Rajesh Babu and Santhosh Kumar Gugulothu, [3] investigated CFD analysis of heat transfer enhancement by using passive technique in heat exchanger. The heat transfer enhancement is very important many engineering applications to increase the performance of heat exchangers. The active techniques required external power like surface vibrations, electrical fields etc and the passive techniques are those which does not required any external power but the inserts are required to disturb the flow like tape inserts etc moreover literature survey says passive techniques gives more heat transfer rate without external power requirement by keeping different tape inserts. However CFD tool is very important and effective tool to understanding heat transfer applications. Computational heat transfer flow modelling is one of the great challenges in the classical sciences. By incorporating the inserts the heat transfer enhancement is increased due to its importance in different applications. By CFD modelling by taking concentric tube by considering with and without inserts we conclude that heat transfer enhancement by using ANSYS Fluent version 14.5.

Praveen Kumar Kanti et al, [4] investigated CFD analysis of shell and tube heat exchanger.Analysing shell and tube heat exchanger without baffle plates by changing their outer material.The calculations and simulations are done for counter flow of the heat exchanger.

Digvendra Singh et al, [5] investigated CFD analysis of shell and tube heat exchanger using different headers. The heat exchanger is designed as per the commercial needs of the industry. Kern's technique is used to design the heat exchanger .The designing procedure results in a shell and tube heat exchanger having 21 tubes, 170mm shell diameter and 610 mm long. As the designing procedure doesn't include the type of the header to be used, so we have analysed three types of header which can provide a uniform velocity in the inlet of each tube. Different geometries are included in different positions of the inlet nozzle for the header. CFD simulations are used for the optimum positioning of the inlet nozzle which could be proposed from the uniform distribution of the liquid methanol and the uniform velocity distribution though each and every tube. The main objective of this paper is to verify the heat exchanger designed with the use of the Kern's technique, by the use of Commercial Computational Fluid Dynamics (CFD) software. For the simulation, purpose a symmetric view of the simplified geometry of the heat exchanger is made using solid works software.

C.K.Pardhi and Dr.PrasantBaredar, [6] investigated performance improvement of double pipe heat exchanger by using turbulator. The objective is to reduce as many of the factor as possible like capital cost, maintenance cost, space and weight, consistence with safety and reliability. This work describes the principal techniques of industrial importance for theaugmentation of single phase heat transfer on the inside of tubes namely twisted tapes. Twisted tape should be used in heat exchanger when high heat transfer rate is required and pressure drop is of no significance.

II. METHODOLOGY

- Geometry Modeling using ANSYS Design Modeler.
- Meshing in Ansys work Bench
- Identification of flow domain
- Specification of boundary Conditions
- Selection of solver parameters
- Convergence criteria
- Results and post processing

III. EXPERIMENTAL: SIMULATION

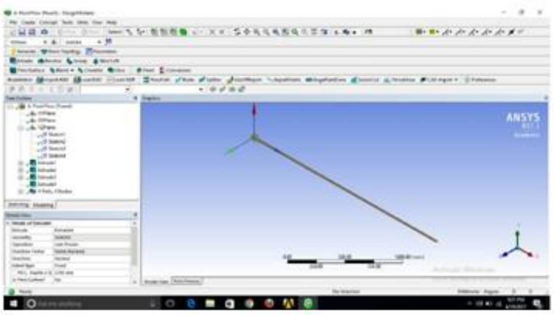
3.1 ANSYS:

Ansys is the finite element analysis code widely use in computer aided engineering (CAE) field. ANSYS software help us to construct computer models of structure, machine, components or System, apply operating loads and other design criteria, study physical response such as stress level Temperature distribution, pressure etc.

In Ansys following Basic step is followed:

1. During pre-processing the geometry of the problem is defined.
2. Volume occupied by fluid is divided into discrete cells (the mesh). The mesh may be uniform or non-uniform.
3. The physical modelling is defined.
4. Boundary condition is defined. This involves specifying the fluid behaviour of the problem. For transient problem boundary condition are also defined.
5. The simulation is started and the equation are solved iteratively as steady state or transient.
6. Finally a post procedure is used for the analysis and visualization of the resulting problem.

The modelling of the Heat Exchanger is done using finet element model available in ANSYS workbench



3.1.1 Meshing:

Initially a relatively coarser mesh is generated with 1.8 Million cells. This mesh contains mixed cells (Tetra and Hexahedral cells) having both triangular and quadrilateral faces at the boundaries. Care is taken to use structured cells (Hexahedral) as much as possible, for this reason the geometry is divided into several parts for using automatic methods available in the ANSYS meshing client. It is meant to reduce numerical diffusion as much as possible by structuring the mesh in a well manner, particularly near the wall region. Later on, for the mesh independent model, a fine mesh is generated with 5.65 Million cells. For this fine mesh, the edges and regions of high temperature and pressure gradients are finely meshed

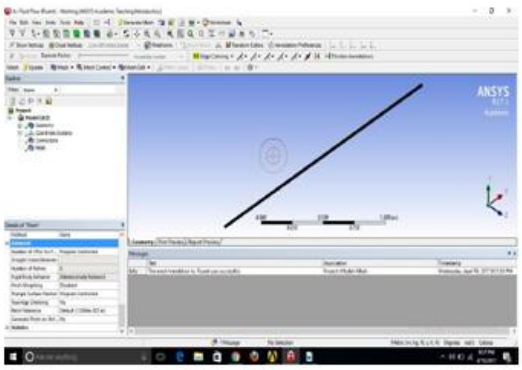


Fig.3.2. Heat exchanger pipe after meshing

3.2 FLUENT SETUP

CFD is a sophisticated computationally-based design and analysis technique. CFD software gives the power to simulate flows of gases and liquids, heat and mass transfer, moving bodies, multiphase physics, chemical reaction, fluid-structure interaction and acoustics through computer modelling. This software can also build a virtual prototype of the system or device before can be apply to real-world physics and chemistry to the model, and the software will provide with images and data, which predict the performance of that design.

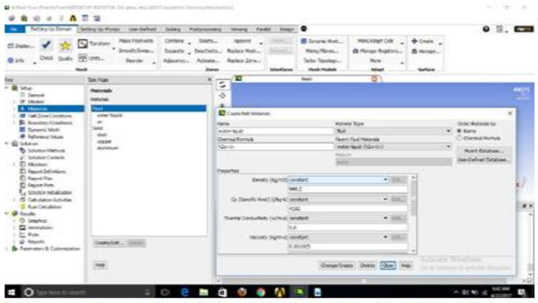


Fig.3.3. Copying of material properties from fluent database

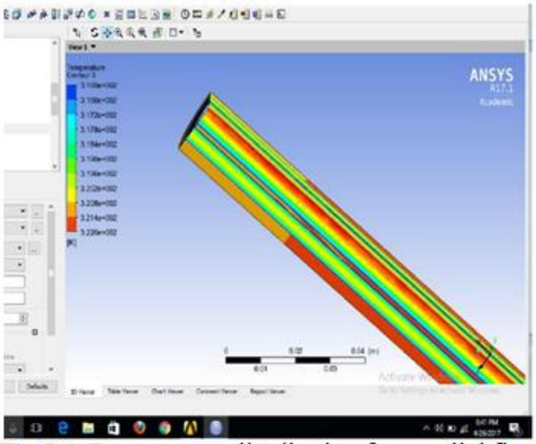


Fig.3.4. Temperature distribution for parallel flow

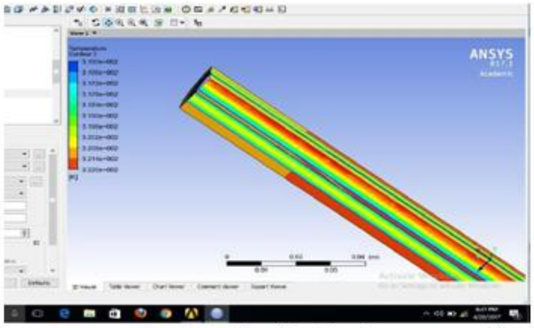


Fig.3.5. Temperature distribution for counter flow

IV. CALCULATIONS

4.1 THEORETICAL CALCULATIONS OF PARALLEL FLOW HEATEXCHANGER

a) **PROCEDURE**

1. The valve on the water main is opened and the water is allowed to circulate through the heat exchanger.
2. Thermometers are inserted at entries and exits of both hot and cold fluids.
3. The electric geyser is switched on and water is heated in it.
4. The valve 5 is opened to allow the hot water from geyser to flow through the Inner copper tube continuously.
5. For parallel flow arrangement, the valves 1 & 2 are opened and valves 3 and 4 are closed, to cause both fluids to flow in the same direction.
6. The cold water valve 1 is controlled to ensure that both fluids are flowing at the same rate (by measuring jar and stopwatch).
7. At steady state, the temperatures and timings for collection of 1000 ml of cold and hot fluid are noted.
8. The experiment is repeated with cold water flow rate twice that of hot water.

b) **OBSERVATIONS:** FOR HOT WATER:
Time taken for 1 litre of water collection th = 24 sec

Inlet temperature Thi = 470 C Outlet temperature Tho = 430 C FOR COLD WATER:
Time taken for 1 litre of water collection t = 24 sec
Inlet temperature Tci = 330 C
Outlet temperature Tco = 370 C Calculations

PARAMETER	VALUE
Mass flow rate of hot water= Mass flow rate of cold water	0.04166 kg/sec
Heat transfer rate of hot water Qh=Heat transfer rate of cold water Qc	0.6978 KW
Overall heat transfer coefficient Uo	0.839 KW/m2 K
Effectiveness	0.286

Table 4.1 Parallel flow heat exchanger

4.2 THEORETICAL CALCULATIONS OF COUNTER FLOW HEAT EXCHANGER

a) PROCEDURE:

1. The valve on the water main is opened and the water to allowed to circulate through the heat exchanger.
2. The electrical geyser is switched on and water is heated in it.
3. The valve 5 is opened to allow the hot water to flow through the inner copper tube Continuously.
4. For counter flow arrangement, the valves 3 & 4 are opened and valves 1 and 2 are closed, to cause both fluids flow in the opposite directions.
5. Thermometers are inserted at entry and exit conditions of both hot and cold fluids.
6. The cold water valve 3 is controlled to ensure both fluids are flowing at the same rate (by measuring jar and stop watch).
7. At steady state the temperatures and timings for cold and hot fluid flow are noted.
8. The experiment is repeated with cold water flow rate twice the hot water flow rate.

b) OBSERVATIONS:
FOR HOT WATER:
Time taken for 1 litre of water collection th = 32 sec

MODEL CALCULATIONS:

PARAMETER	VALUE
Mass flow rate of hot water	0.03125 kg/sec
Mass flow rate of cold water	0.0625 kg/sec
Heat transfer rate of hot water Qh=Heat transfer rate of cold water Qc	1.308 KW
Overall heat transfer coefficient Uo	0.8980 KW/m2 K
Effectiveness	0.5

Table 4.2 counter flow heat exchanger

V. EXPERIMENTALVALUES OBTAINED FROM THE CFD ANALYSIS

5.1 For parallel flow:
FOR HOT WATER :
Time taken for 1 litre of water collection th = 24 sec

Inlet temperature Thi = 470 C Outlet temperature Tho = 440 C FOR COLD WATER :
Time taken for 1 litre of water collection = 24 sec
Inlet temperature Tci = 330 C Outlet temperature Tco = 360 C

c) MODEL CALCULATIONS:

PARAMETER	VALUE
Mass flow rate of hot water= Mass flow rate of cold water	0.04166kg/sec
Heat transfer rate of hot water Qh=Heat transfer rate of cold water Qc	0.523 KW
Overall heat transfer coefficient Uo	0.5404 KW/m2 K.
Effectiveness	0.21

Table 5.1 Parallel flow heat exchanger

5.2. For counter flow:
FOR HOT WATER:
Time taken for 1 litre of water collection th = 32sec
Inlet temperature Thi = 550 C
Outlet temperature Tho = 470 C FOR COLD WATER:
Time taken for 1 litre of water collection = 16 sec
Inlet temperature Tci = 330 C
Outlet temperature Tco = 37.50 C **Model Calculations:**

PARAMETER	VALUE
Mass flow rate of hot water	0.03125 kg/sec
Mass flow rate of cold water	0.0156 kg/sec
Heat transfer rate of hot water Qh=Heat transfer rate of cold water Qc	1.0467 KW
Overall heat transfer coefficient Uo	.7651 KW/m2 K
Effectiveness	0.45

Table 5.2 Counter flow heat exchanger

VI. RESULT

Fig.6.1 pressure distribution over pipe

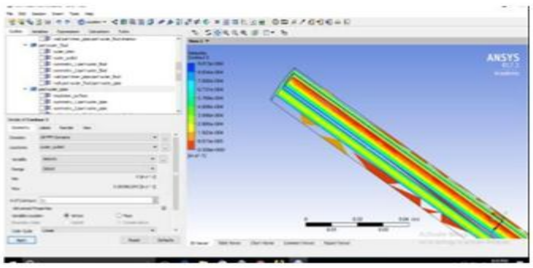


Fig.6.2 Velocity distribution over pipe

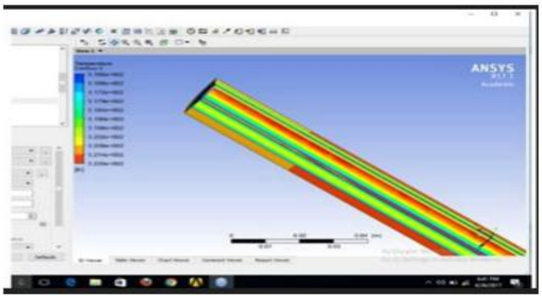


Fig.6.3 Temperature distribution of parallel flow

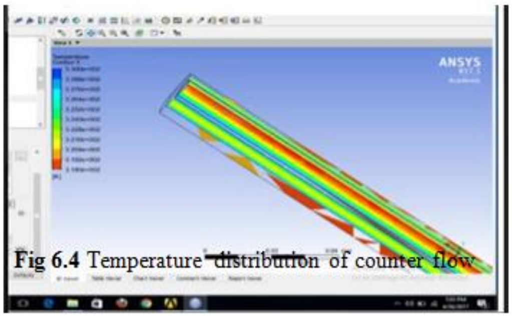


Fig 6.4 Temperature ~~distribution~~ of counter flow

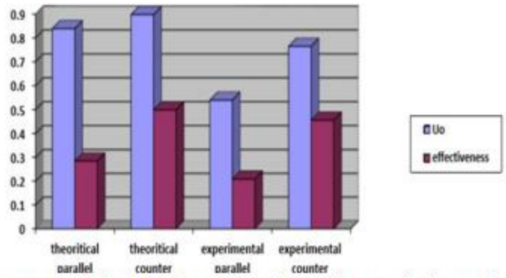


Chart 61. Above chart describes the variation of parameters

VII. CONCLUSIONS

The performance, CFD analysis of different fluids and different pipe materials were investigated on parallel and counter flow in concentric tube het exchanger. The conclusions of the investigating at are as follows.

- The main objective of this project was to analyses the fluid flow in double pipe heat exchangers and the subsequent performance of these heat exchangers.
- Design the double pipe heat exchanger by using ANSYS WORKBENCH.
- To simulate heat transfer in concentric tube heat exchanger by using CFD-Fluent software.
- The ANSYS FLUENT results were found to be fairly consistent with hard calculations with most of the values within 5% of each other

REFERENCES

[1]. D.Bhanuchandrarao, M.Ashokchakravarthy, Dr. Y. Krishna, Dr. V .V. SubbaRao, T.HariKrishna "CFD Analysis And PerformanceOf Parallel And Counter Flow In Concentric Tube Heat Exchangers", ISSN: 2278-0181,Vol. 2 Issue 11, November - 2013
[2]. Nice Thomachan, Anoop.K.S, Deepak.C.S, Eldhose.P.Kuriakose, HabeebRahman.K.K,Karthik.K.V"CFDanalysis of tube in tube heat exchanger with fins",

e-ISSN: 2395 - 0056, p-ISSN: 2395-0072, Volume:03 Issue: 04 | Apr-2016

[3]. CRajeshBabuandSanthosh KumarGugulothu "CFD analysis of heat transfer enhancement by using passive technique in heat exchanger", International Journal ofRecent advances in Mechanical Engineering (IJMECH) Vol.4, No.3, August 2015

[4]. Praveen Kumar Kanti, Karthika.U.P, Sabeer Ali, SanathKumar.N, ShyamChandran C"CFD analysis of shell and tube heatexchanger", ISSN: 2319-6890)(online),2347-5013(print) VolumeNo.5Issue:Special 6, pp: 1129 -1254, 20 May 2016

[5]. Digvendra Singh, Dr.JitendraPandey andDr. Abhishek Tiwari "CFD analysis of shell and tube heat exchanger using different header section", ISSN (ONLINE): 2321-3051, Vol.4 Issue 4, April 2016

[6]. C.K.Pardhi,Dr.PrasantBaredar"Performanceim provement of double pipe heat exchanger byusingturbulator",ISSN:2250-3676, Volume-2, Issue-4, 881-885

[7]. White, F.M., Fluid Mechanics, 3rd edition. Mc- Graw Hill, 1994

[8]. Frank P. Incropera, David P. Dewitt, TheodoreL.Bergman,AdrienneS.Lavine"Fund amentals Of Heat And Mass Transfer", John Wiley & Sons SIXTHEDITION, 111 River Street, Hoboken, NJ 07030-5774 2007.

Miss Sneha.H.Dhoria "Cfd Analysis on Concentric Tube Heat Exchanger in Parallel and Counter Flow Direction "International Journal of Engineering Research and Applications (IJERA) , vol. 8, no.6, 2018, pp.20-25