IARJSET

ISSN (Online) 2393-8021
ISSN (Print) 2394-1588

International Advanced Research Journal in Science, Engineering and Technology
National Conference on Design, Manufacturing, Energy & Thermal Engineering (NCDMETE-2017)
AGTI's Dr. Daulatrao Aher College Engineering, Vidyanagar Extension, Karad
Vol. 4, Special Issue 1, January 2017

Numerical Simulation of Shell and Tube Heat Exchanger by using CFD

G.A. Yadav[1], S.V. Janugade[2], M.R. Patil[3]

Assistant Professor, Mechanical Engineering, AGTI's Dr. Daulatrao Aher College of Engineering, Karad, India [1]

Assistant Professor, Mechanical Engineering, AGTI's Dr. Daulatrao Aher College of Engineering, Karad, India [2]

U.G. Student, Mechanical Engineering, Dr. J.J. Magdum College of Engineering, Jaysingpur, India[3]

Abstract: Heat transfer is the terms use for thermal energy transfer from a hot to a colder body. Heat transfer is considered as transfer of thermal energy from physical body to another. Heat transfer is the most important parameter to be which is measured as the performance and efficiency of the shell and tube heat exchanger. CFD is the science of predicting fluid flow, heat transfer, mass transfer, chemical reactions, and related phenomenon by solving the mathematical equations which govern these processes using a numerical process. By using CFD Simulation software, it can reduces the operation cost as well as time compared by Experimental in order to measure the optimum parameter and the behaviour of this type of heat exchanger, conclusion from this project work is using CFD analysis instant results can be obtained. Temperature at any moment or point can be procured during the process. Flow patterns can be visualized which is not possible in case of experimental analysis. Also internal geometry is visible.

Keywords: Heat Exchanger, CFD, ANSYS, Fluent.

INTRODUCTION

Heat transfer always occurs from a hot body to a cold one, which is a result of the second law of thermodynamics. Theoretically on a microscopic scale, thermal energy is related to the kinetic energy of molecules. The greater a material's temperature, the greater the thermal agitation of its constituent molecules. Then the regions containing greater molecular kinetic energy will pass this energy to regions with less kinetic energy. So when a physical body likes an object or fluid, is at a different temperature than its surroundings or another body, heat transfer will occurs in such a way that the body and the surroundings reach thermal equilibrium.[1]

Heat transfer always occurs from a hot body to a cold one, a result of the second law of thermodynamics. Where there is a temperature difference between objects in proximity, heat transfer between them can never be stopped but can only be slowed down. Transfer of thermal energy can only occurs through three ways which is conduction, convection and radiation or any combination of that.[2]

The ultimate goal of the field of computational fluid dynamics (CFD) is to understand the physical events that occur in the flow of fluids around and within designated objects. These events are related to the action and interaction of phenomena such as dissipation, diffusion, convection, shock waves, slip surfaces, boundary layers and turbulence .In the field of aerodynamics, all of these phenomena are governed by the compressible Navier – Stokes equations .Many of the most important aspects of these relations are nonlinear and, as a consequence, often have no analytical solution. This, of course, motivates the numerical solution of the associated partial differential equations. At the same time it would seem to invalidate the use of linear algebra for the classification of the numerical methods [1].

LITERATURE REVIEW

The purpose of this chapter is to provide a literature review of past research effort such as journals or articles related to shell and tube heat exchanger and computational fluid dynamics (CFD) analysis whether on two dimension and three dimension modelling. Moreover, review of other relevant research studies are made to provide more information in order to understand more on this research.[2] It is debatable as to who did the earliest CFD calculations (in a modern sense) although Lewis Fry Richardson in England (1881-1953) developed the first numerical weather prediction system when he divided physical space into grid cells and used the finite difference approximations of Bjerknes's "primitive differential equations". His Own attempt to calculate weather for a single eight-hour period took six weeks of real Time and ended in failure! His model's enormous calculation requirements led Richardson to propose a solution he called the "forecast-factory". The "factory" would have involved filling a vast stadium with 64,000 people. Each one, armed with a Mechanical calculator would perform part of the flow calculation. A leader in the center, using colored signal lights and telegraph communication, would coordinate the forecast. What he was proposing would have been a very rudimentary CFD calculation. CFD is now recognized to be a part of the computer-aided engineering (CAE) spectrum of tools used extensively today in all industries, and its approach to modeling fluid

IARJSET

ISSN (Online) 2393-8021
ISSN (Print) 2394-1588

International Advanced Research Journal in Science, Engineering and Technology

National Conference on Design, Manufacturing, Energy & Thermal Engineering (NCDMETE-2017)

AGTI's Dr. Daulatrao Aher College Engineering, Vidyanagar Extension, Karad

Vol. 4, Special Issue 1, January 2017

flow phenomena allows equipment designers and technical analysts to have the power of a virtual wind tunnel on their desktop computer. CFD software has evolved far beyond what Navier, Stokes or Da Vinci could ever have imagined. CFD has become an indispensable part of the aerodynamic and hydrodynamic design process for planes, trains, automobiles, rockets, ships, submarines; and indeed any moving craft or manufacturing process that mankind has devised (Fluent.com)[1]

Design of shell and tube heat exchanger in ANSYS Geometry

Heat exchanger geometry is built in the ANSYS 14.5 workbench design module. Geometry is simplified by considering the plane symmetry and is cut half vertically.

Table1: Heat Exchanger Dimensions

No.	Description	Value (mm)
1.	Drum Length	800
2.	Shell diameter	250
3.	Tube outer diameter	32
4.	Tube inner diameter	26
5.	Number of tubes Nt	14
6.	Shell/Tube length L	800
7	Tube bundle geometry and pitch Triangular	35
8	Central baffle spacing, B	266.66
9	Number of baffles. Nb	2

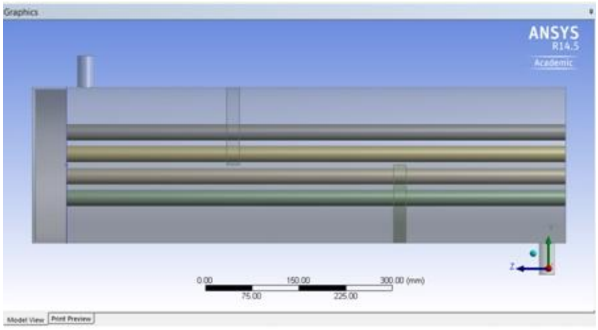


Image 1: Complete model of shell and tube heat exchanger

Mesh

Initially mesh is generated with 1.8 Million cells. This mesh contains mixed cells i.e. Tetra and Hexahedral cells having both triangular as well as quadrilateral faces at the boundaries. Care is taken to 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 by 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, mesh contains 2.2 million cells.

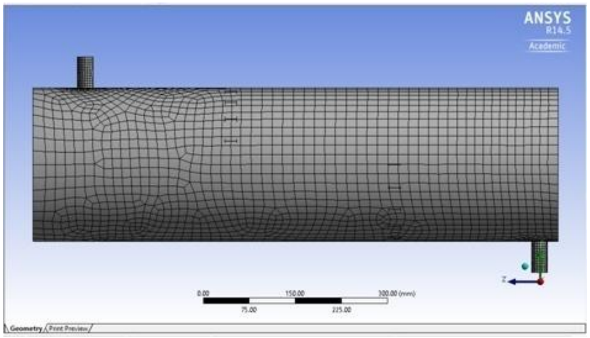


Image 2: Meshing diagram of shell and tube heat exchanger

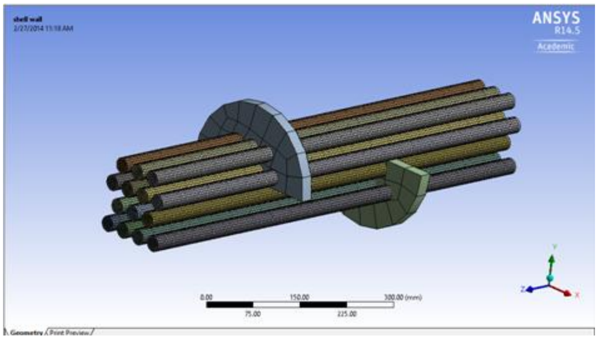


Image 3: Meshing diagram of tubes in shell and tube heat exchanger

Boundary conditions

Boundary condition remains the simple and cheap form to prescribe, when compared with other theoretically more satisfying selections in CFD.Boundary conditions are also required to be assigned for external stationary solid wall boundaries that are used to bind the flow geometry and the surrounding walls of possible internal obstacles within the flow domain.

Table2: Definition of boundary type for entity

Entity(Edge)	Type
Tube Inlet	Mass Flow Inlet
Tube Outlet	Mass Flow Inlet
Shell Inlet	Mass Flow Inlet
Shell Outlet	Pressure outlet
Baffles	Wall
Drum(Cylinder)	Wall

Table3: The Co-ordinates of the edges and respective labels for geometry

Sr. No	Co- ordinates	Label
1.	(0.0,0.0,0.0)-(0.0.,0.0,0.0)	Tube Inlet
2.	(0.0,0.0,0.0)-(0.0,800,0.0)	Tube Outlet
3.	(0.0,0.0,0.0)-(140,20,)	Shell Inlet
4.	(0.0,0.0,0.0)-(140,780,)	Shell Outlet

ISSN (Online) 2393-8021
ISSN (Print) 2394-1588

IARJSET

International Advanced Research Journal in Science, Engineering and Technology

National Conference on Design, Manufacturing, Energy & Thermal Engineering (NCDMETE-2017)

AGTI's Dr. Daulatrao Aher College Engineering, Vidyanagar Extension, Karad

Vol. 4, Special Issue 1, January 2017

5.	1.(0.0,0.0,0.0)-(0.0,266.66,10,) 2.(0.0,0.00.0) (0.0,533,10)	Baffles
6.	(0.0,0.0,0.0)-(250,800,250)	Drum(Cylinder)

CFD simulations

- The total time required for a flow simulation depend on
- The choice of numerical algorithms and data structures
- Linear algebra tools, stopping criteria for iterative solvers
- Discretization parameters such like mesh quality, mesh size, time step.
- Cost per time step and convergence rates for outer iterations
- Programming language (most CFD codes are written in FORTRAN)
- Many other things like hardware, vectorization, parallelization etc. The quality of simulation results depends on
- The mathematical model and underlying assumptions
- Approximation type, stability of the numerical scheme
- Mesh, time step, error indicators, stopping criteria

ANSYS Fluent

The union of ANSYS Fluent into ANSYS Workbench provides users with superior bi-directional connections to all major CAD systems, powerful geometry modification and creation with ANSYS design modeller technology, and advanced meshing technologies used in ANSYS Meshing. The platform also allows data and results to be shared between applications using an easy drag-and-drop transfer.

X-Y Plots

Plots are mainly two-dimensional graphs which are represents the variation of one dependent transport variable against another independent variable. They can usually be drawn by hand or many more conveniently plotting packages. Such X-Y plots are the most definite and quantitative way to present the numerical data.

Vector Plots

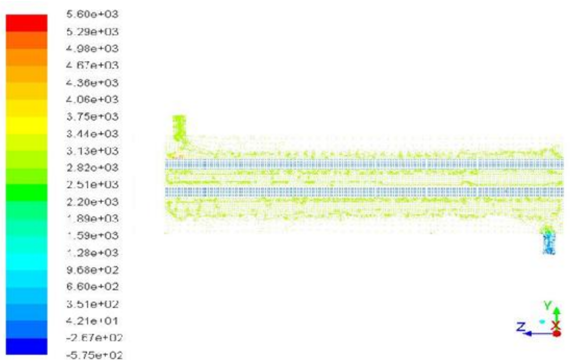


Image 5: Velocity Vectors Colour by Static Pressure.

A vector plot provides a vector quantity is displayed at discrete points (usually velocity, with arrows) whose orientation demonstrates direction and whose sizes demonstrate magnitudes. It generally represents a perspective view of the flow field in two dimensions. In a three dimensional flow of field, different slices of two dimensional planes containing the vector quantities can be made in different orientations to better scrutinize the global flow phenomena. If the mesh densities are considerably high, the CFD user can either interpolate or reduce the numbers of output locations to avoid the clustering of these arrows "obliterating" the graphical plot.

Contour Plots

In CFD, contour plots are one of the most commonly found graphic representation of data. A contour line (also known as isoline) can be described as a line indicative of some property that in constant in space. The equivalent representation in three- dimensional is an iso-surface. In contrast to X-Y plots, contour plots like vector plots provide a global description of the fluid flow enclose in one view. Generally, contours are plotted such that the difference between the numerical value of the dependent transport variable from one contour line to the neighbouring contour line is taken as constant. The use of contour plots is usually not targeted for precision evaluation of the numerical values between contour lines. Although some mental and/or numerical interpolation can be performed between contour lines in space, it is at the very least an imprecise process. The actual numerical values represented by the isolines of these plots are some times less important than their overall disposition. [2]

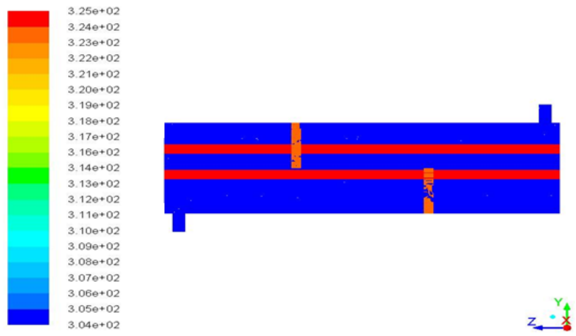


Image 6: Counters of Total Temperature (k)

RESULT AND DISCUSSION

The most important in design of experimental system is that the arrangement must confirm, in terms of boundary condition, flow configuration, and the ambient condition, to the physical problem being studied. In addition, measurements have to be planned so as to justify the applicability of experimental system with respect to the process under study.

Component Specifications

The set up for the rig is shown in Images 7 and 8.The set up consists of following components:

IARJSET
ISSN (Online) 2393-8021
ISSN (Print) 2394-1588
International Advanced Research Journal in Science, Engineering and Technology
National Conference on Design, Manufacturing, Energy & Thermal Engineering (NCDMETE-2017)
AGTI's Dr. Daulatrao Aher College Engineering, Vidyanagar Extension, Karad
Vol. 4, Special Issue 1, January 2017

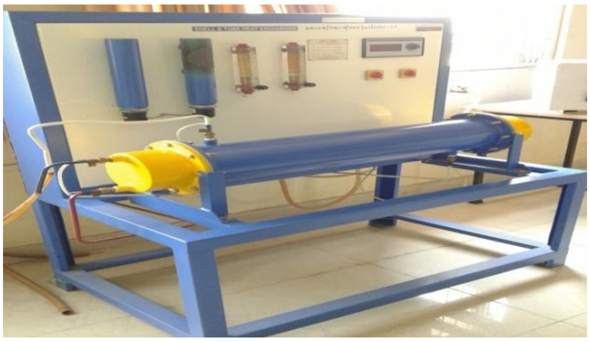


Image 7


Image 8

1. Cylinder (Shell)
Material: M.S, Size: ϕ 250 × 800 mm (6mm thick)
2. Tubes
Number of tubes: 14, Size: ϕ26× 800 mm. (6 mm thick)
3. Baffles
Number of Baffles: 2
Size: ϕ 250 (half circular, 10mm thick)
4. Inlet/Outlet of Shell
Number of Inlet and Outlet: 1 each, Size: ϕ 26×15 mm
5. Pitch
Triangular pitch of 35mm.
6. Rhotameter
Capacity: 1.1 to11 lpm.
7. Gyesors
2KW, 2Nos, for hot water supply
8. Make: Data cone enterprises pvt.lmt.Sangaliwadi, Sangali.
9. Manufacture year: 2010-2011.

Experimental Set up
The horizontal cylinder made up of M.S is clamped at bottom .The tubes are placed horizontally in the cylinder. Cylinder having openings (i.e. inlet and outlet) at opposite sides to each other, but tubes having inlet and outlet on same side.

The four thermocouples are attached at inlet and outlet of shell and tube resp., the temperatures are sensed by these thermocouples is averaged and provided to computer software for further processing. The software used for further processing is ANSYS ICEM, FLUENT. The rhotameter and temperature indicator are used to take the readings of input water flow rate and temperature.

Formulae's are used
Heat exchanger effectiveness ($\mathcal{E}_H$)
This is the ratio of actual heat transfer to the maximum possible heat transfer [5]

$$\mathcal{E}_H = \frac{Q}{Qmax} = \frac{Th1 - Th2}{Th1 - Tc1}$$

Effective logarithmic mean temperature difference (LMTD)
This is the mean temperature difference between the two fluid streams [5]

$$P = \frac{Tc2 - Tc1}{Th1 - Th2}$$

Where P = ratio of cold fluid temperature difference and hot fluid temperature difference [5]

$$R = \frac{Th1 - Th2}{Tc2 - Tc1}$$

Where R = ratio of hot fluid temperature difference and cold fluid temperature difference. The correction factor, F is estimated from charts using calculated value of P and R

Overall heat transfer coefficients, U
The required overall heat transfer coefficient is given as [5]

$$U_{req} = \frac{q}{AF(\Delta TM)}$$

Where, q = Heat transfer rate in the heat exchanger,
A = Surface area, F = correction factor of logarithm mean temperature difference,
ΔTM= Logarithmic mean temperature difference.

Ɛ-NTU Method
Heat transfer unit (NTU)[5]

$$NTU = \frac{UoAo}{(mcp)min}$$

Capacity ratio

$$Cr = \frac{(mcp)min}{(mcp)max}$$

Set of Readings Taken On Experimental Set Up

Table7: Readings Taken On Experimental Set Up

Hot water flow rate (Q_H) (lpm)	Cold water flow rate (Q_C) (lpm)	Hot water Inlet temp. $T_{Hi}(T_1)$ {°C}	Hot water Outlet temp. $T_{Ho}(T_2)$ {°C}	Cold water Inlet temp. $T_{Ci}(T_3)$ {°C}	Cold water Outlet temp. $T_{Co}(T_4)${°C}
3	2.5	52	47	31	36

ISSN (Online) 2393-8021
ISSN (Print) 2394-1588
IARJSET
International Advanced Research Journal in Science, Engineering and Technology
National Conference on Design, Manufacturing, Energy & Thermal Engineering (NCDMETE-2017)
AGTI's Dr. Daulatrao Aher College Engineering, Vidyanagar Extension, Karad
Vol. 4, Special Issue 1, January 2017

3.6	2.5	52	46	31	41
4.1	2.5	52	45	31	42
4.8	2.5	52	47	31	45
5.3	2.5	52	48	31	47

Calculation for 1st reading:-

Heat Transfer from hot water (q_h)

$q_h = M_h \times C_{ph}(T_{Hi}-T_{Ho})$

= 1.9876 Watt.

Heat Transfer from cold water (q_c)

$q_c = M_c C_{pc}\ (T_{Co}-T_{Ci})$

= 1.9170Watt.

Average heat transfer rate (q_o)

$q_o = \frac{qh+qc}{2}$

= 1.9533Watt.

Logarithmic Mean Temp. Difference (L.M.T.D.)[5]

$$\Delta T_m = \frac{\Delta Ti-\Delta To}{\ln\left(\frac{\Delta Ti}{\Delta To}\right)}$$

$$= \frac{21-11}{\ln(21/11)}$$

= 13.904°K

Overall Heat Transfer Coefficient[5]

$q = U_o \times A_o \times \Delta T_m$

$A_o = \Pi d_o L$

= 0.0804

U_o = 1.7473Watt/m°k

Effectiveness $= \frac{qo}{(Cmin\ (Thin - Tcin\ ))}$

= 0.238

Efficiency $= \mathcal{E} * \frac{Surface\ \ Area}{Total\ \ Surface\ \ Area}$

= 35.41%

Experimental Result Table:-

Table 8: Experimental Result

Sr. No.	Hot water flow rate (Q_H) (lpm)	L.M.T.D.	Overall heat transfer coefficient(U_o) (Watt/m°K)	Average heat transfer rate(q_o) (Watt)	Effectiveness (Ɛ)	Efficiency (η) (%)
1.	3	13.904	1.7473	1.9533	0.238	35.41
2.	3.6	12.896	2.024208	2.09877	0.2857	42.5
3.	4.1	11.888	2.3669	2.2623	0.33	49.58
4.	4.8	14.448	1.9667	2.2846	0.2380	35.41
5.	5.3	15.451	0.85534	2.3049	0.1904	28.33

Observation table of Heat Exchanger by Using CFD

Table9: Observation table of Heat Exchanger by Using CFD

Hot water flow rate (Q_H)(lpm)	Cold water flow rate (Q_C)(lpm)	Hot water Inlet temp. $T_{Hi}(T_1)$ {°C}	Hot water Outlet temp. $T_{Ho}(T_2)${°C}	Cold water Inlet temp. $T_{Ci}(T_3)$ {°C}	Cold water Outlet temp. $T_{Co}(T_4)$ {°C}
3	2.5	52	49	31	40
3.6	2.5	52	50	31	42
4.1	2.5	52	51	31	44
4.8	2.5	52	49	31	43
5.3	2.5	52	49	31	45

Result Table of Heat Exchanger by using CFD:

Table10: Result Table of Heat Exchanger by using CFD

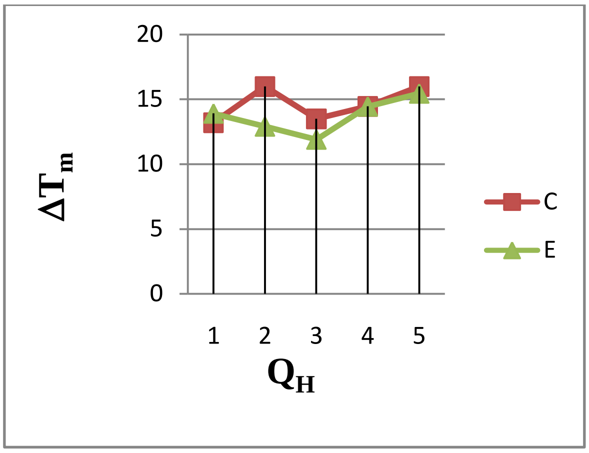
Sr. No.	Hot water flow rate (Q_H) (lpm)	L.M.T.D.	Overall heat transfer coefficient (U_o) (Watt/m°K)	Average heat transfer rate(q_o) (Watt)	Effectiveness (Ɛ)	Efficiency (η) (%)
1.	3	13.1919	1.824895	1.9355	0.1904	28.33
2.	3.6	15.9791	1.63374	2.099	0.2857	42.5
3.	4.1	13.4938	2.137604	2.3191	0.3809	56.67
4.	4.8	14.4481	1.95667	22.2729	0.2380	35.42
5.	5.3	15.9791	1.794073	2.3049	0.1904	28.33

IARJSET
ISSN (Online) 2393-8021
ISSN (Print) 2394-1588
International Advanced Research Journal in Science, Engineering and Technology
National Conference on Design, Manufacturing, Energy & Thermal Engineering (NCDMETE-2017)
AGTI's Dr. Daulatrao Aher College Engineering, Vidyanagar Extension, Karad
Vol. 4, Special Issue 1, January 2017

Graphs

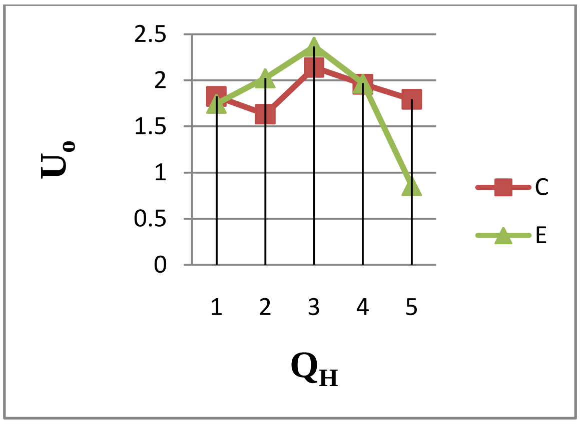
E = Experimental results , C = CFD results.

1. Hot Water Flow Rate Vs L.M.T.D.:-



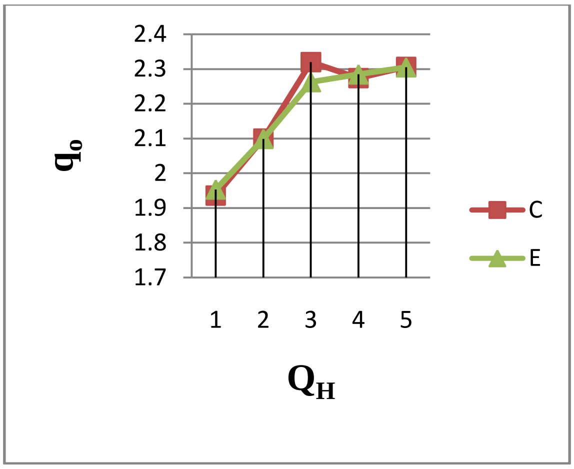
Graph 1

2. Hot Water Flow Rate Vs Overall Heat Transfer Coefficient:-



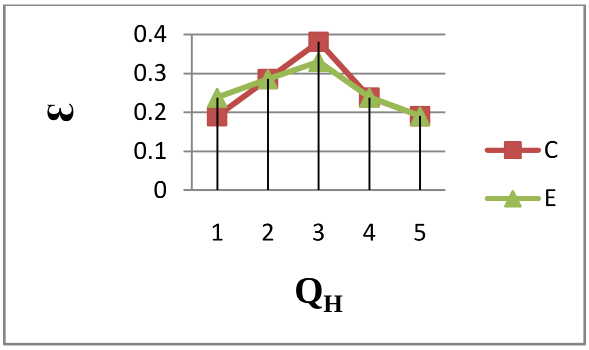
Graph 2

3. Hot Water Flow Rate Vs Average heat Transfer Rate:-



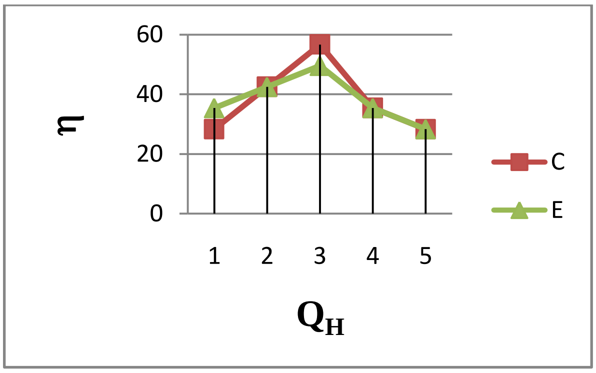
Graph 3

4. Hot Water Flow Rate Vs Effectiveness:-



Graph 4

5. Hot Water Flow Rate Vs Efficiency:-



Graph 5

CONCLUSION

The flow visualization on experimental setup is a costlier affair. It involves the use of heavy and expensive machines. Since this process is a costly and time consuming this gives motivation to carry out the CFD analysis of the experimental setup of heat exchanger as a project work.

- Using CFD analysis instant results can be obtained.
- Temperature at any moment or point can be procured during the process.
- Flow patterns can be visualized which is not possible in case of experimental analysis.
- Also internal geometry is visible.

REFERENCES

[1] John D Anderson "Computational Fluid Dynamics-The Basic with Application." , McGraw- Hill, New York, 1995, pp 2-50.
[2] Jiyuan Tu, Guan Heng Yeoh , Chaoqun Liu ," Computational Fluid Dynamics A Practical Approach , Elsevier , First edition 2002, pp 1-61.
[3] Yunus A. Cengel, "Heat and Mass Transfer", Second Edition, 2002, pp17-32,459-514.
[4] Jian-Fei Zhang, Ya-Ling He, Wen-Quan Tao, "3d numerical simulation of shell and tube heat exchanger with middle-overlapped helical baffle", a journal, School of energy and power engineering, china.
[5] Raj put R.K. (2009), "Heat and Mass Transfer", Ram-Nagar, New Delhi.