International Journal of Engineering and Techniques - Volume 3 Issue 6, Nov - Dec 2017

RESEARCH ARTICLE OPEN ACCESS

CFD Analysis of Shell and Tube Heat Exchanger for Heat Transfer Capabilities

Mr.Santosh K Katarki[1],Mr. Anandkumar S Malipatil[2]

[1]Dept of Mechanical Engineering, PG Student VTU RC Kalaburagi

[2],Dept of Mechanical Engineering,Asst. Professor VTU RC Kalaburagi

Abstract:

Heat exchangers are more often than not used equipments in the process industries. All most all process industries use heat exchangers for cooling, heating, condensation or boiling purpose. A heat exchanger is simply a device or a mechanical device which is designed expeditiously in order to transfer or exchange heat from one substance to another. If two fluids are alleged at altered temperatures heat exchangers are used to exchange the heat between them. In this paper efforts have been made to design a heat exchanger by modelling in ANSYS software which is having a inner diameter of 330mm and outer diameter of 350mm for shell. Similarly for tube inner diameter is 21.18mm and outer diameter 25.4mm, length of tube is 1500mm, contains 36 tubes. Here assembly of shell and tube are done with water and steam as a medium. By using results the design can be altered for better efficiency. Realizable k–ε (RKE) model plies first-rate results.

Keywords **— Shell & Tube Heat exchanger,** Ansys 14.5, **k–ε (RKE) model, Temperature, Pressure**

I. INTRODUCTION

Heat exchangers are more often than not used equipments in the process industries. All most all process industries use heat exchangers for cooling, heating, condensation or boiling purpose.In the developing technology, we are using many heat exchangers to exchange the heat from one matter to another matter. A heat exchanger is simply a device or a mechanical device which is designed expeditiously in order to transfer or exchange heat from one substance to another. If two fluids are alleged at altered temperatures heat exchangers are used to exchange the heat between them.

In this project efforts have been made to design a heat exchanger by modelling in ANSYS software which is having a inner diameter of 330mm and outer diameter of 350mm for shell . Similarly for tube inner diameter is 21.18mm and outer diameter 25.4mm, length of tube is 1500mm and number of tubes are 36. Here assembly of shell and tube are done with water and steam as a medium. By using results the design can be altered for better efficiency.A diagrammatic representation of counter flow type heat exchanger is shown below.

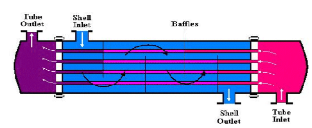

Figure 1.1: Counter-flow Heat Exchanger Arrangement

II.LITERATURE SURVEY

To converge this work, literature survey in depth is carried out. The summary of literature survey for some of the references is mentioned as follows.

Nakka Sita Rama Raju. V Jaya Prasad,S Kamaluddin,A Sunny Kumar[2017].

"In this paper a modified model of shell and tube type heat exchanger which is having both interacting mediums as water and steam is designed. In this paper we have first designed a shell and tube heat exchanger to heat water from 40℃ to 83℃ by steam at 140℃ temperature. This plan has been done using Kern's method in order to gain various dimensions such as shell, tubes, baffles etc. A computer model using CATIA V5 has been built up by using the derived dimensions of heat exchanger. After that the thermal simulation in ANSYS has been executed by applying several thermal loads on different faces and edges. The heat transfer capacious of more than a few thermal materials have been equated by assigning different materials."

[2016]Abdulsalam D.Mohamed, Aamer M.Al-Dabagh, Duaa A.Diab.

"The Heat pipe heat exchanger (HPHE) is considered as a one of the most useful appliance for the revival of waste heat energy. An observational study has been carried out on air to – air HPHE constructed of thermosyphon heat pipes with distilled water as the working fluid and a fill ratio of 75% from the evaporator length. Its model was composed of 4 rows, each row contains 12 copper tubes, each tube have ID= 9.5 mm, OD=10mm and length =950 mm and the rows of tubes were coiffured in a staggered manner. Aluminum wavy plate fins of 0.1mm thickness were fixed among the tubes to increase the heat transfer area. Different tests were conducted at various flow rates (air flow rate through evaporator and condenser sections) ranged between 0.12 and 0.37 kg/s and at dissimilar temperatures of air entering evaporator section (90, 100,110) to specify disconsolately in the effectiveness when the flow rate ratio equal to one . Theoretical model based on (ε-NTU) is devastated by using visual basic language computer program to scrutiny of the temperature distribution along heat pipe heat exchanger. The results from this model were likened with experimental results. A relationship between the experimental and theoretical results shows little discrepancy".

Raj Kumar Yadav M Veena Nayak Jain, Vikas Mukhraiya[2016].

"Always Engineers are recurrently being demanded to progress processes and amplify efficiency. This appeal may occur as a result of the need to increase process throughout, increase profitability, or put up capital limitations. Processes which use heat transfer equipment must habitually be improved for these reason. This paper provide the application of Taguchi method in heat exchanger performance."

III.METHODOLOGY

Basically CFD analysis involves three major steps say Pre-Processing, Processing and Post Processing.

1. Pre-Processing: This is preliminary step of CFD simulation process which assists in explaining geometry in good suitable manner. The flow domain of interest is divided into equal number of smaller parts known as elements. There are different Pre-Processing software available are Gridgen, CFD-GEOM, ANSYS Meshing, ANSYS ICEM CFD, T Grid etc. Pre-processing this includes defining the problem, creating the 3D model, fetching the model to Ansys workbench, meshing, and applying physical operating condition called boundary conditions.

2. Solving or Processing: Once the issue material science has been recognized, liquid properties, stream physical science model, limit circumstances are located to tackle utilizing PC. There are renowned business programmings accessible for this including: CFD++, Open FOAM, ANSYS CFX, Star CCM, ANSYS FLUENT and so on. All the above given programming have their individual abilities. Utilizing this product it is conceivable to fathom the administering conditions identified with stream material science issue. Handling includes unraveling of numerical or scientific conditions of liquid stream until the point when joining in result is accomplished. Typically it requires the PC to understand a huge number of conditions and may take couple of hours to few days.

3 Post processing: The last stride in the wake of getting the outcomes from the solver is to examine the outcomes with various techniques like weight and speed shape plots, vector plot, streamlines, temperature contour and so forth At the point when the model has been comprehended, the outcomes can be broke down both numerically and graphically. Post-preparing is about perception either in basic 2-D to 3-D portrayals.

IV.FLOW CALCULATION

"Flow is controlled by using continuity equation, energy equation and Navier-Stokes momentum equations. Transport of mass, energy and momentum occur through convective flow and diffusion of molecules and turbulent eddies. All equations are set up over a control volume".

Governing equations:

3-D equation of continuity for Unsteady state:-

$$\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u)}{\partial x} + \frac{\partial (\rho\, v)}{\partial y} + \frac{\partial (\rho\, w)}{\partial z} = 0$$

Equation for Momentum:-

$$u\frac{\partial u}{\partial x} + \vartheta\,\frac{\partial u}{\partial y} = \frac{\mu}{\rho}\,\frac{\partial^2 u}{\partial y^2}$$

Equation of Energy:-

$$\rho Cp\left(\frac{\partial T}{\partial t} + u\,\frac{\partial T}{\partial x} + v\,\frac{\partial T}{\partial y} + w\,\frac{\partial T}{\partial z}\right)\emptyset + \frac{\partial}{\partial x}\left[k\,\frac{\partial T}{\partial x}\right] + \frac{\partial}{\partial y}\left[k\,\frac{\partial T}{\partial y}\right] + \frac{\partial}{\partial z}\left[k\,\frac{\partial T}{\partial z}\right] + u\,\frac{\partial p}{\partial x} + v\,\frac{\partial p}{\partial y} + w\,\frac{\partial p}{\partial z}$$

$\emptyset$ – Dissipation function which is given by the following equation.

$$\emptyset = 2\mu\left[\left(\frac{\partial u}{\partial x}\right)^2 + \left(\frac{\partial v}{\partial y}\right)^2 + \left(\frac{\partial w}{\partial z}\right)^2 + 0.5\left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\right)^2 + 0.5\left(\frac{\partial v}{\partial z} + \frac{\partial w}{\partial y}\right)^2 + 0.5\left(\frac{\partial w}{\partial x} + \frac{\partial u}{\partial z}\right)^2\right] - \frac{2}{3}\,\mu\,\left(\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z}\right)^2$$

μ - Viscosity, p-pressure, Cp-Specific heat, T-Temperature, ρ - density, (x,y,z)- coordinate, u,v,w- velocity components.

V.CFD ANALYSIS

a. Geometry

A software model is accomplished by using proportions of shell, tubes and baffles in ANSYS 14.5.

Workbench design modeler is used to construct Heat exchanger geometry and for further analysis. Geometry is made easy by making an allowance for the plane symmetry. This heat exchanger is counter flow type and the tube side consists of one inlet and one outlet representing of 36 complete tubes and taking into consideration the symmetry. After model is generated, run thermal simulation of heat exchanger. The outcomes prevailed were moderately well-known with general circumstance.
The parts independently as well as in congregation are as shown below:

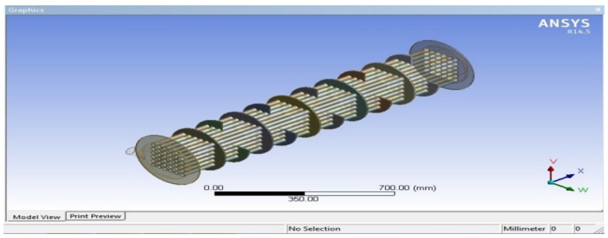

Fig 5.1. Tube and Baffel assembly

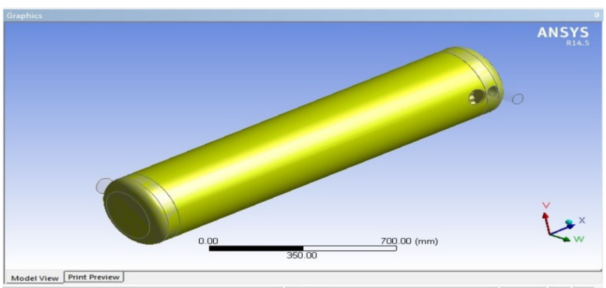

Fig 5.2 Tube, Baffle, Header & shell assembly

b. Mesh Generation

To execute finite element analysis the mesh is generated. Conciliation between computer speed and mesh quality is made in this process.
The model of mesh generated is shown in fig:

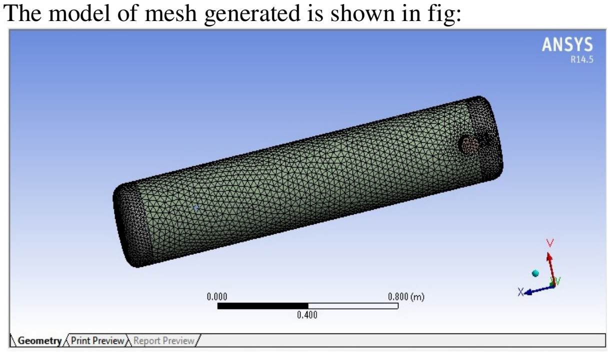

Fig 5.3 Mesh Model

VI. TURBULENCE MODEL

From the mean-square vorticity fluctuation, Dissipation rate (ε) equation is derived which is fundamentaly diverse from the SKE.
For Reynolds stresses quite a few reliability surroundings are enforced.
Reimbursement:
1. Planar and round jets Spreading rate are anticipated flawlessly.
2. Flows involving rotation, boundary layers under strong adverse pressure gradients

VII. BOUNDARY CONDITIONS:

Boundary conditions are defined:
Inlet – Velocity inlet
Outlet – pressure outlet (zero guage pressure)
Walls – convective heat transfer, Stationary, no slip condition.
Fluid Material – Water and Water vapor
Solid Material – copper and Steel
Energy equation – ON
Turbulence Model - K-epsilon, realizable, Standard wall treatment model
Solution Method – Second Order
Initialization from Inlet condition

VIII. VALIDATION & SOLUTION GRAPHS

The solution converge for twenty iterations .

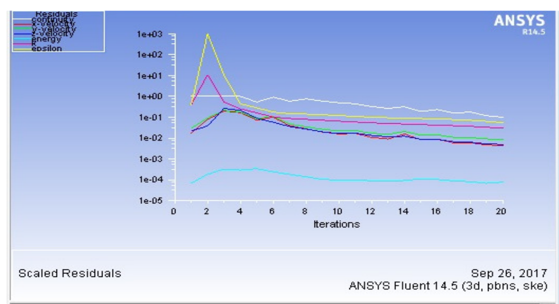

Fig 8.1. 20 iteration graph

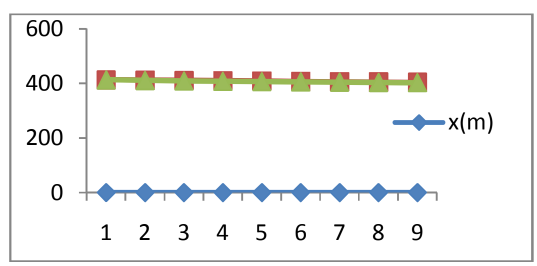

Fig 8.2 Graph of Temperature v/s length for hot fluid

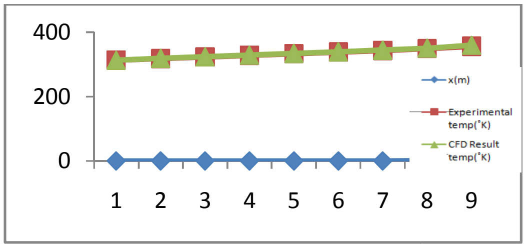

Fig 8.3 Graph of Temperature v/s length for cold fluid

IX RESULTS AND DISCUSSION:

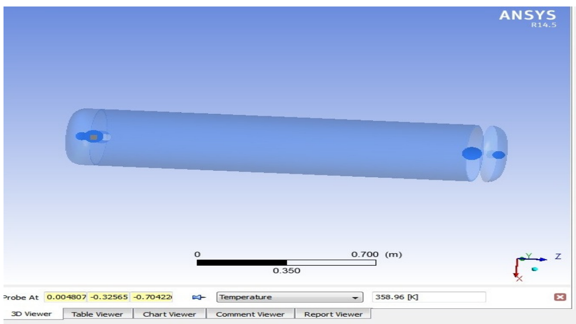

Fig 9.1 Cold Outlet Temperature

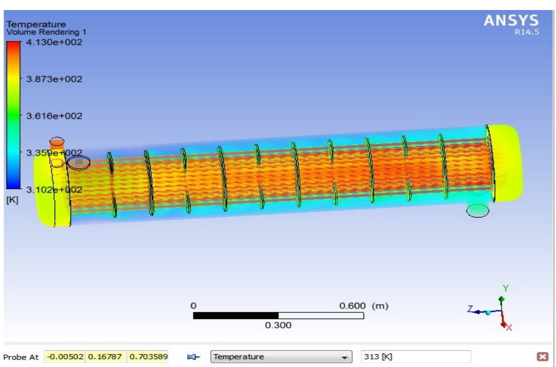

Fig 9.2 Cold Inlet Temperature

it is clear, temperature of cold fluid increases because of 37.5% baffale cut compared to 25.Which increases Heat transfer rate.

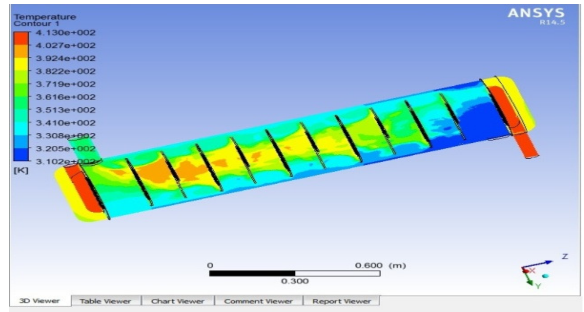

Fig 9.3 Temperature Contour1 on Y-Z plain

The above fig shows temperature distribution at different length by various colors on y-z plane due to heat exchange

.

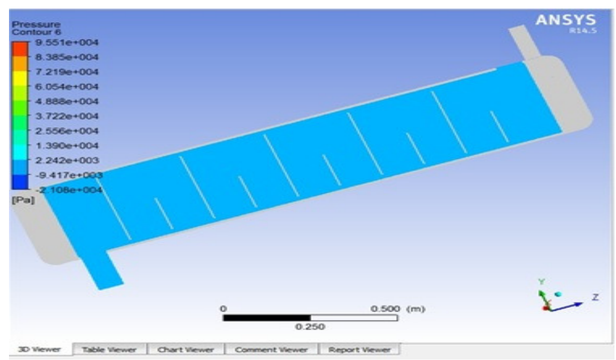

Fig 9.4Pressure Contour on Y-Z plain

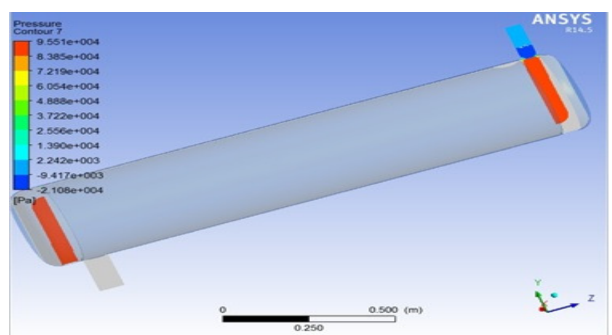

Fig 9.4Pressure Contour

Pressure decreases from 95512-21076 Pa from inlet to outlet.

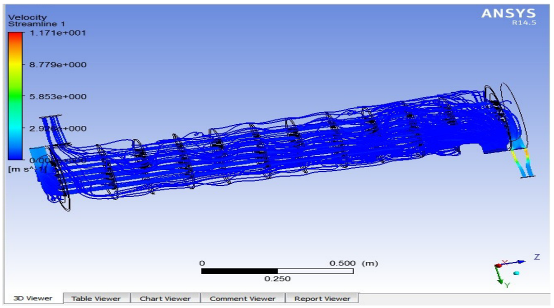

Fig 9.5 Streamline velocity contour

Reynolds number is above 2,50,683 for cold ,10269121 for hot fluid. Stream line flow is turbulent.

X CONCLUSION

It is clear that if we increase the shell side area so that mass flow rate of cold fluid increases enhancing heat transfer rate and apportioning copper(cu) to the whole assemblage, hence for above discussed materials best possible value of heat transfer rate is obtained; nevertheless that will also be a very pricey affair. It is concluded that temperature of cold fluid increased from 356 to 359 ºK. Hot fluid decreases 404 - 403 ºK. Accomplished value of heat transfer rate for hot fluid is 11475.76 KW and cold fluid is 64221.52 KW. Consequently the comparisons of temperatures of experimental value to the attained CFD value are valid.
Pressure values are Inlet-95512 Pa, outlet- (-21076)Pa. Turbulent KE (J/kg) varies from 4 at inlet to 0 at outlet. Stream line &Vector velocity values are 0-12m/s.

X FUTURE SCOPE

By using Reynolds Stress Models, the model is ameliorated, but computational costs requisite is high. Sometimes fluid present at shell side get around the tube bundle without interaction, hence heat transfer is ascertained to be poor. Thus better heat transfer can be done in 2 ways: To keep the outer fluid mass flux lower, shell diameter is reduced OR to enhance the inner fluid mass flux tube, spacing can be increased.Area is reduced to enlarge the inner fluid (Shell side) mass flux. By producing cross-flow regions design can further be improved such that flow doesn't keep on parallel or counter to tubes. It will routinely increase heat transfer by permiting the outer shell fluid to jumble with the inner shell fluid.

REFERENCES

[1] V Jaya Prasad , "Thermal analysis of shell and tube type Heat exchanger to demonstrate the Heat transfer capabilities of various Thermal materials using ansys" vol 8,issue 2 (2017).

[2] J.P. Holman, Heat Transfer, Tata Mc Graw Hill international publishers 9th edition,2002

[3] Donald Q Kern, Process Heat Transfer, Tata Mc Graw Hill international publishers edition 1997,

[4] Glittle flower, s.vani latha,G.sahaya Baskaran, "Investigation on the role of nanosilver in controlling",International, Journal of Engineering Research ISSN: (2319-6890) (online), 2347- 5013(print) Volume No.3 Issue No: Special 1, pp: 21-25

[5] Raj Kumar Yadav and Veena Nayak Jain, "Thermal Analysis of Heat Exchanger with the Help of Taguchi Method". *International Journal of Advanced Research in Engineering and Technology*, 7(1), 2016, pp. 01-06

[6] Migbar Assefa Hawassa University, Institute of Technology, Ethiopia,Global Journal of Researches in Engineering: A Mechanical and Mechanics Engineering, Volume 14 Issue 4 Version 1.0 Year 2014 Type: Double Blind Peer Reviewed International Research Journal Publisher: Global Journals Inc. (USA) Online ISSN:2249-4596Print ISSN:0975-5861

[7] Mohammed, K. A., Yisa, G. L., Sule Emmanuel And 4Makwin, H. L.,IOSR Journal of Mechanical and Civil engineering (IOSR-JMCE) e-ISSN: 2278-1684,p-ISSN: 2320-334X, Volume 11, Issue 3 Ver. VI (May- Jun.2014), PP 08-17