ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

Design and Analysis of a Tubular Heat Exchanger Using Computational Fluid Dynamics Method

Pappala Santosh Saikumar
Department of Mechanical Engineering,
Sri Vaishnavi College of Engineering, Srikakulam,
Andhra Pradesh 532001, India.

Sattaru Suresh Babu
Department of Mechanical Engineering,
Sri Vaishnavi College of Engineering, Srikakulam,
Andhra Pradesh 532001, India.

Kotipatruni Ratnakara Rao
Department of Mechanical Engineering,
Sri Vaishnavi College of Engineering, Srikakulam,
Andhra Pradesh 532001, India.

ABSTRACT:

Shell and tube heat exchangers are the most common type of heat exchangers used in present scenario. Heat exchangers are widely used equipment in various industries such as power generation and transportation, refrigeration industry and chemical process industries because it suits high pressure application. The shell and tube heat exchanger is considered in which hot water is flowing inside one tube and cold water runs over that tube. Computational fluid dynamics technique which is a computer based analysis is used to simulate the heat exchanger involving fluid flow, heat transfer.

CFD resolve the entire heat exchanger in discrete elements to find the temperature gradients, pressure distribution and velocity vectors. In this project design of heat exchanger by using ANSYS 15.0 software. Computational Fluid Dynamics is done according to experimental setup. The shell and tube type heat exchanger have been designed and analyzed in the C.F.D. software. By checking hot outlet, cold outlet and heat transfer values with experimental and numerical data. Analysis is done using CFD and calculated values for stream line, temperature contour and velocity analysis for both hot and cold fluid. The heat transfer will be compared with experimental and Numerical data.

INTRODUCTION:

A Heat Exchanger is a device used for transferring the heat between two fluids that are at different temperatures.

Heat Exchangers are useful in many engineering processes like those in refrigerating and air conditioning systems, power systems, food processing systems, chemical reactors and space or aeronautical applications and dairy applications. Heat moves in a heat exchanger includes convection in every liquid furthermore, conduction through the divider isolating the two liquids. In the present investigation of heat exchanger, it is helpful to work with a general heat exchange coefficient 'U' that records for the impact on heat exchange. The rate of heat exchange between the two liquids of heat exchanger relies upon the extent of the temperature contrast at that area, which shifts along with the heat exchanger. In the examination of heat exchangers, it is typically advantageous to work with the logarithmic mean temperature distinction (LMTD), which is a comparable mean temperature contrast between the two liquids for the whole heat exchanger.

APPLICATIONS OF HEAT EXCHANGER:

A heat exchanger is a device used to transfer heat between on or more fluid and fluid may be separated by solid wall to prevent them from mixing. Heat exchangers are widely used in industry for cooling and heating large scale industrial processes. In industry there is huge waste of energy due to exhaust gas, these gases can be recovered by the use of heat exchanger.

Cite this article as: Pappala Santosh Saikumar, Sattaru Suresh Babu & Kotipatruni Ratnakara Rao, "Design and Analysis of a Tubular Heat Exchanger Using Computational Fluid Dynamics Method", International Journal & Magazine of Engineering, Technology, Management and Research, Volume 4, Issue 12, 2017, Page 66-77.

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

This can save a lot of energy and money of the industry by using the exhaust gases in the industry. It can also be used in many industries including waste water treatment, refrigeration, wine and beer making, nuclear power, petroleum refining. Double type heat exchanger and plate type heat exchanger are used in waste water treatment to maintaining optimal temperature. Heat exchanger is also used in air craft to take heat from the engine's oil system to heat cold flue. Due to this fuel efficiency improves as well as reduces the possibility of water entrapped in the fuel freezing.

LITERATURE REVIEW:

Rehman [1] has designed a un- baffled shell and tube heat exchanger with respect to heat transfer coefficient and pressure drop. The flow and temperature field inside the shell and tube have resolved using CFD package considering plane symmetry. He has done CFD simulation for a single and tube bundle and compared with the experimental results. Temperature and velocity profile are examined and it is found that flow remains parallel to the tubes thus limiting the heat transfer. 2/3rd of the shell side fluid is bypassing the tubes and little bit contribution to the overall heat transfer coefficient. Due to cross flow and higher temperature difference tube and shell side fluid the higher heat flux is observed at shell's inlet.

By using Reynold stress model with higher computational costs modeling can be improved. The heat transfer is found to be poor because the most of the shell side fluid by-passes the tube bundle without interaction. Thus the design can be modified in order to achieve the better heat transfer in two ways. Either, the shell diameter is reduced to keep the outer fluid mass flux lower or tube spacing can be increased to enhance the inner fluid mass flux. Design can further be improved by creating cross flow region in such a way that flow does not remain parallel. Venkatesh et al. [2] have observed characteristics of spiral flow heat exchanger at various Reynold no and base temperature.

They also use the concept of thermo electric effect for direct conversion of temperature difference to electric voltage for the measuring of temperature at various points in the heat exchanger. For developing the model they use CAD and for simulation CFD are used. ANSYS CFX is used for input boundary condition which is considered for experimental purpose. They show that the temperature of the base plate decreases as the mass flow rate is increased and finally heat transfer rate has increased due to mass flow rate. Pressure profile shows that at the inlet it is more and at outlet it is less it's because of velocity. In velocity profile, it can be seen that the velocity increases with increase in mass flow rate. This is mainly due to two reasons: Due to turbulence created in the centre of the setup and other reason is due to centrifugal force created as the fluid circulates in circular motion. There is a clear evidence of the heat transfer cross the vertical strip separating the adjacent counter flow hot and cold steams.

For constant inlet and base temperatures, the outlet temperature of the water decreases with mass flow rate. Velocity suddenly increases at mid length due to change in cross section area of SFHE, velocity recovers in a short distance in the downstream and remains almost constant till the outlet. There is no evidence of the influence of the base temperature on the velocity distribution. Also, in experimental setup, the temperature in the centre portion is more. And there is difference in voltage at different points of the setup. This shows that heat transfer is taking place. Also when the voltages are compared to the calibrated graph, the temperature obtained is approximately equal to that obtained computationally. Patil et al. [3] have presented PHE (plate heat exchanger) which is an important type of a condensing or evaporating system. The heat transfer and pressure drop is most important part during sizing and rating the performance of PHE. CFD can be a tool for future to ensure effective design of plate heat exchangers. As in process industry every day operating parameters change with different processes, it is difficult to design without higher

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

reliability factor or oversize design. Ravaged et al. [4] have studied to measure the performance of different designs; its model is suitably designed and fabricated so as to perform experimental tests. Thermal analysis has been carried out for different design with base fluids and on the basis of comparative results is made which one gives the best heat transfer rates. This study shows the design and thermal analysis of different tubes. Experimentally, same designs are made and results are evaluated. With relate to same design tubes are thermally analyzed in ANSYS software. The possibility to increase in these characteristics using the latest technology and various methods has raised application range of these designs. Modified design tubes are having great applications due to their large heat transfer area and high heat transfer coefficients. The main applications are used in many industrial processes like waste water treatment, refrigeration, wine, petroleum refining and beer making.

OBJECTIVE:

The objective of this experiment is to find out the heat transfer rate of shell and tube type heat transfer. With the help of using ANSYS tool 15.0 for designing and simulation of heat exchanger setup with vertical baffles. For more advancement visit the lab for experiment and familiarize with the experimental setup with the consent of the teaching assessment.

EXPERIMENTAL ANALYSIS

2.1 DETAILS OF EXPERIMENTAL SET UP:

The operation of shell-and-tube heat exchanger involves the consideration of both convective and conductive heat transfer. The shell and tube heat exchanger consists of number of tubes is parallel enclosed in a cylindrical shell. Heat is transferred between one fluid flowing through the tubes and another fluid flowing through the cylindrical shell around tubes. Baffles are included inside the shells to increase the velocity of the fluid to improve the heat transfer. The exchanger is designed to demonstrate liquid to liquid heat transfer.

In the present research the author is considered Shell and tube heat exchanger (one shell and tube 12 tubes with two transverse baffles in the shell) to analysis the heat transfer rate. The determination of the overall heat transfer coefficient is one of the most important, and often most uncertain, quantities in the analysis of heat exchangers. This coefficient primarily accounts for all of the conductive and convective resistance (k and h, respectively) between fluids separated by the wall (or tube), and further takes into account thermal resistances caused by fouling on the wall (i.e., rust, scaling, etc.) by means of fouling factors on both sides. Another important quality in heat exchanger analysis is the total rate of heat transfer between the hot and cold fluid. Several different expressions are this heat transfer rate can be developed, relating the heat transfer coefficient. When these expressions are developed, care must be taken to ensure that the appropriate mean temperature expressions are used. Several assumptions can be made to simplify these expressions. In this, we assume negligible heat transfer between the system and surroundings, negligible potential or kinetic energy changes, constant specific heats, and these fluids are not undergoing any phase change. In this case, the total heat transfer rate, Q, becomes

$$\varrho_h = m_h C_{ph} (T_{hi} - T_{ho})$$

$$\varrho_c = m_c C_{pc} (T_{ci} - T_{co})$$

When the total heat transfer rate is rated to the overall heat transfer coefficient, another expression develops. In this time, where A is the area for heat transfer and LMTD is the log mean temperature difference between the inlet and outlet temperatures. In a shell-and tube heat exchanger, the area for heat transfer is

$$U = \frac{\varrho}{A \times LMTD}$$

2.2 TUBES:

Since the desired heat transfer in the exchanger takes place along the tube surface, the selection of the tube geometrical variables is important from the performance point of view. In most of the applications, plain tubes are used. However, the additional surface area is required to compensate for low heat transfer coefficient on shell side. The most common plain tube sizes are 15.88,19.05, and 25.40mm (5/8, 3/4, and 1in.) tube outside diameters. From the heat transfer viewport, smaller diameter tubes yield higher heat transfer coefficients and result in a more compact exchanger. However, large-diameter tubes are easier to clean and more rugged. The foregoing common size represents a compromise. For mechanical cleaning, smallest practical size is 19.05mm (3/4 in.). For chemical cleaning, small sizes can be used to provide that the tubes never plug completely.

2.3 BAFFLES:

Baffles are commonly placed in the shell to force the shell-side fluid to flow across the shell to enhance heat transfer and to maintain uniform spacing between the tubes. Despite their widespread use, shell and tube heat exchangers are not suitable for use in automotive and aircraft applications because of their relatively large size and weight. Note that the tubes in a shell-and tube heat exchanger open to some large flow areas called headers at both ends of the shell, where the tube-side fluid accumulates before entering in the tubes and after leaving them.

Type of Baffles:

a. Plate Baffles

b. Rod Baffles

c. Impingement Baffles

- **PLATE BAFFLES:**

Plate Baffles are two types. One is disk and doughnut. Single and double segmental baffles are used frequently. The single segmental baffles are generally recommended. The practical range of single segmental baffle spacing is 1/5 to 1 shell diameter, although optimum could be 2/5 to ½. The minimum baffle spacing for cleaning the bundle is 50.8 mm (2 in.) or 1/5shell diameter, whichever is larger. Spacing closer than 1/5 shell diameter provides added leakage that nullifies the heat transfer advantage of closer spacing. This type of plate baffles have been used in the experimental setup.

2.4 SHELLS:

The shell, the most common due to low cost and relative simplicity, it is used for single phases hell fluid applications and for small condensers with low pressure volumes. Multiple passes onthe tube side increases the heat transfer coefficient h. However, a multi pass tube arrangement scan reduce the heat exchanger effectiveness or F factor compared to that single-pass arrangement(due to some tubes passes being in parallel flow) if the h is increased and NTU do not compensate for the parallel flow effect. The two E shells in series may be used to increase the exchanger effectiveness.

Front-End Heads:

The front end is stationary, while the rear end head can be either stationary or floating depending on the allowed thermal stresses between tubes and the shell. The major criteria for the selection of front-and rear-end heads are maximum thermal stresses, operating pressure, easy to clean, free from hazards and cost effectiveness.

Rear-End Heads:

In a shell-and-tube heat exchanger, the shell is a temperature different from that of tubes because of heat transfer between the shell and the fluids. These results are a differential thermal expansion and stress among the shell, tubes, and the tube sheet. If the proper provisions are not made, the shell or tubes can buckle or tubes can be pulled apart out of the tube sheet. Provision is made for different thermal expansion in the rear-end-heads.

2.5 INSULATION:

Thermal insulations are materials or combinations of materials that are used primarily to provide resistance to heat flow.

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

You are probably familiar with several kinds of insulations available in the market. Most of the insulations are heterogeneous materials made of low thermal conductivity materials, and they involve air pockets. This is not surprising since air has one of the lowest thermal conductivities and is readily available. The Styrofoam commonly used as a packaging material for TVs, VCRs, computers, and just about anything because of its lightweight is also an excellent insulator. Temperature difference is the driving force for heat flow, and the greater temperature difference, the larger rate of heat transfer. We can slow down the heat flow between two mediums at different temperatures by putting "barriers" on the path of heat flow. Thermal insulations serve as such barriers, and they play a major role in the design and manufacture of all energy efficient devices or systems, and they are usually the cornerstone of energy conservation projects. In the experimental analysis of shell and tube type process, we are analyzing different fluid flow rate and mass flow.

Tubes, Shell, Baffles, Hot water inlet and Outlet, Cold water inlet and outlet Temperature points are the parts are used in experimental setup. In this type of heat exchanger, we used the parallel flow. Tube in this setup used is made of copper material and 12 number of tubes have been used in the setup as shown in the figure shown below tube side is of cold water. Shell is made of mild steel material and the shell side is of cold water. Three numbers of Baffles are used in between tubes to flow the fluid and for the holding purposes of the tubes. Insulation is used in this setup. It is covered the shell and tubes. The total amount of the fluid takes place into the inner space of the insulation. By the heater it is heated for some minutes. After heating, the fluid is allowed to flow through inlet and outlet tubes. The fluid flowing from inlet and outlet of the tubes we will take the readings simultaneously both the hot and cold fluid in 500 ml time for collecting water in sec, temperature will display on setup, mass flow rate of hot water and cold water and overall heat transfer coefficient.

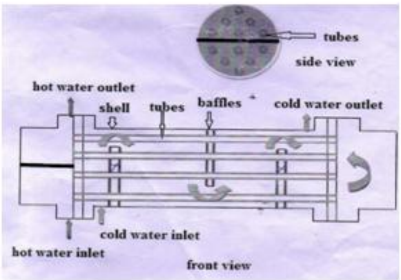

Figure Sectional view of heat exchanger of experimental setup

2.6 SPECIFICATION AND DIMENSIONS:

- Type of heat exchanger: Parallel flow
- No of Baffle : 3nos.

Tube:

- Material : Copper
- Diameter :10mm
- Pitch :30mm(triangular pitch)
- Length :180mm
- No of tubes :12nos.
- Tube side :Hot water

Shell:

- Material : Mild steel
- Diameter :150mm
- Length :200mm
- Shell side :Cold water

Temperature Points

- Cold fluid Inlet (T1)
- Cold fluid Outlet (T2)
- Hot fluid Inlet (T3)
- Hot fluid Outlet (T4)
- Heater capacity : 3kw

Table 1: Heat transfer rate recorded using experimental setup

Temperature C				Time for collecting 500ml of Cold water in sec	Time for collecting 500ml of Hot water in sec	Mass flow rate of cold water kg/s	Mass flow rate of hot water kg/s	Overall heat transfer coefficient, $U_{W/m^2 \cdot K}$	
T_1 (Ci)	T_2 (Co)	T_3 (Hi)	T_4 (Ho)					Shell Material	Tube Material
25.6	44.8	73.4	50.6	41.04	38.37	0.012183	0.01303	504.04	930.99

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

2.7 MATHEMATICAL CALCULATION

▪Heat Rejected in Hot Water (Qh):

$$\varrho_h = m_h C_{ph} (T_{hi} - T_{ho})$$

= 0.013031×4183(73.4-50.6)= 1242.79watts

▪Heat absorbed by Cold Water (Qc):

$$\varrho_c = m_c C_{pc} (T_{ci} - T_{co})$$

=0.012183×4178× (44.8-26.5)= 931.48watt

▪ Logarithmic Mean Temperature Difference

(LMTD):

$\boldsymbol{\theta_1} = (\mathbf{T_{hi}} - \mathbf{T_{ci}})$=(73.4-26.5)=46.9c

$\boldsymbol{\theta_1} = (\mathbf{T_{ho}} - \mathbf{T_{co}})$=(50.6-44.8)=5.8c

$$\text{LMTD} = \frac{\boldsymbol{\theta_2 - \theta_1}}{\ln(\boldsymbol{\theta_2 - \theta_1})} = \frac{(5.8-46.9)}{\ln\left[\frac{5.8}{46.9}\right]} = 19.66^0\text{C}$$

▪Overall Heat Transfer Coefficient (UO):

For shell material (used material=Mild steel)

$$U_o = \frac{\varrho}{A_0 \times \text{LMTD}} \text{ in } w/m^2$$

=931.48/0.094×19.66

=504.04 w/m^2 c

For Tube Material (used Copper)

$$U_{tube} = \frac{\varrho}{A_{tube} \times \text{LMTD}} \text{ in } w/m^2$$

=1242.79/0.0679×19.66

= 930.99 w/m^2 c

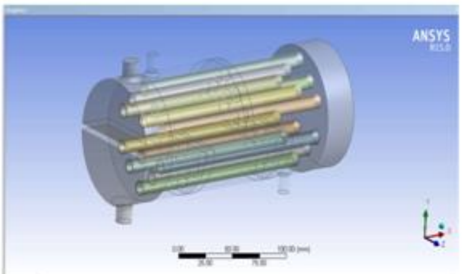

Figure 2 Isometric view of arrangement of baffles and tubes of shell and tube heat Exchanger

The complete geometry of CFD model of shell and tube type heat exchanger is shown in Figure5. The shell side is used mild steel and tube is used copper material. Hot fluid is flowing throughtubes inside and cold fluid is flowing inside the shell side of heat exchanger. The dimensions aregiven in Table 2

Table 2 Shell and Tube type heat exchanger dimensions are considered for CFD analysis

SHELL	TUBE
Material = mild steel	Material = copper
Diameter = 150mm	Diameter = 10mm
Length = 200mm	Pitch = 30mm (Triangular)
Shell side = Cold water	Length = 180mm
No. of baffles = 3	No. of tubes = 12
	Tube side = Hot water

Grid Generation:

The three-dimensional model is then discredited in ICEM CFD. In order to capture both the thermal and velocity boundary layers the entire model is discredited using hexahedral mesh elements which are accurate and involve less computation effort. Fine control on the hexahedral mesh near the wall surface allows capturing the boundary layer gradient accurately. The entire geometry is divided into three fluid domains Fluid_Inlet, Fluid_Shell and Fluid_Outletand three solid domains namely Solid_Baffle1 to Solid_Baffle3 for three baffles respectively.

The heat exchanger is discredited into solid and fluid domains in order to have better control over the number of nodes. The fluid mesh is made finer then solid mesh for simulating conjugate heat transfer phenomenon. The first cell height in the fluid domain from the tube surface is maintained at 100 microns to capture the velocity and thermal boundary layers. The discredited model is checked for quality and min determinant of 4.12. Once the meshes are checked for free of errors and minimum required quality it is exported to ANSYS.

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

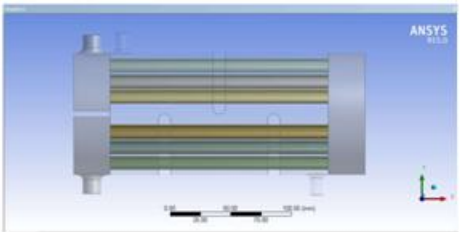

Figure 6 complete model of shell and tube type heat exchanger

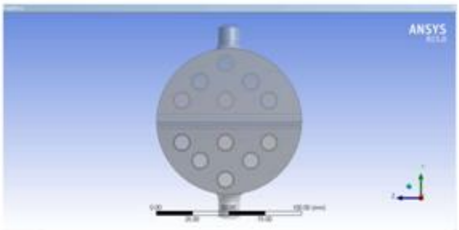

Figure 7 geometrical side view of shell and tube type with showing no. of tubes.

In the above figure shows the total 12 numbers of tubes are attached inside the shell with that arepasses through the three numbers of baffles. In these tubes hot water is flowing. The materialused for the tube is cupper whose thermal conductivity is high it means that heat transfer will bemore.

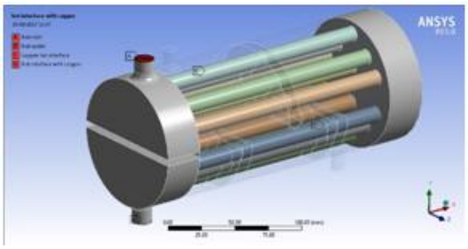

Figure 8 CFD model showing inlet and outlet of fluid flow.

In the model of shell and tube type heat exchanger there are four numbers of parts are showing namely A,B,C and D which is given in Figure 8. A stand for hot inlet from where hot fluid enter the tube. B stands for hot outlet fluid coming out through tubes. C stand for inlet for cold fluids enter the shell. D stands for cold outlet. In the above figure of CFD model showing the baffles used in the heat exchanger analysis. There are three numbers of baffles are used.

The main purpose of using baffles is to give support the tube and flow of fluid direction.

3.3 MESHING:

Initially a relatively coarser mesh is generated with 94995 nodes and 53166 elements with maximum layers are 5 and growth rate is 1.2. 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 94995 nodes. For this fine mesh, the edges and regions of high temperature and pressure gradients are finely meshed.

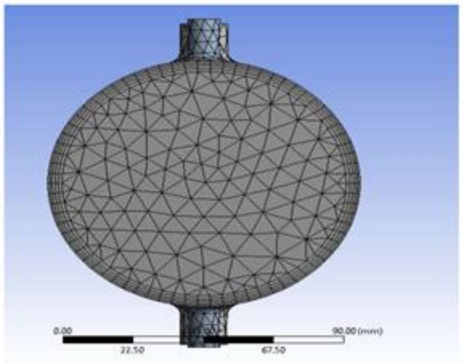

Figure 9 CFD meshing part of tube side view

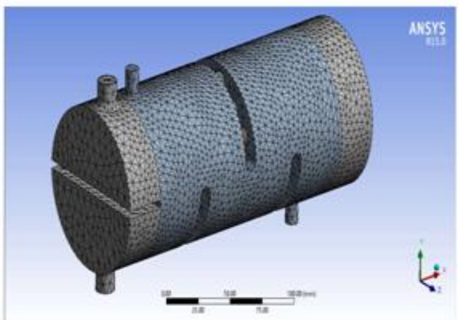

Figure 10 3-D Meshing diagram of shell and tube heat exchanger.

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

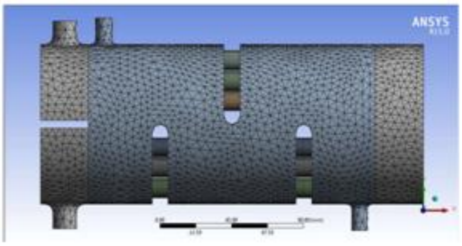

Figure 11 3-D Meshing diagram of shell and tube heat exchanger

TABLE 3 DETAILS OF MESHING:

Object type	Mesh
Minimum edge length	7.598e-003m
Transition ratio	0.272
Growth rate	1.2
Nodes	94995
Elements	53166
Target	41 faces
Relevance centre	Fine

3.4 PROBLEM SETUP:

Simulation was carried out in ANSYS® CFX® v15. In the CFX solver Pressure Based type was selected absolute velocity formation and steady time was selected for the simulation .In the model option energy calculation was on and the viscous was set as standard k-e, standard wall function (k-epsilon 2 eqn). In cell zone fluid water-liquid was selected. Water-liquid and cupper, was selected as materials for simulation. Boundary condition was selected for inlet outlet. In inlet and outlet 0.012183kg/s mass flow rate and temperature was set at 297.35k. Across each tube0.013031kg/s mass flow rate and 343.15k temperature was set. Mass flow was selected in each inlet. In reference Value Area set as 1m2, Density 998 kg/m3 enthalpy 229485j/kg, length 1m, Velocity 1.44085m/s, Ration of specific heat 1.4 was considered. Temperature and mass flow rate are varying in time of both hot fluid and cold fluid respectively.

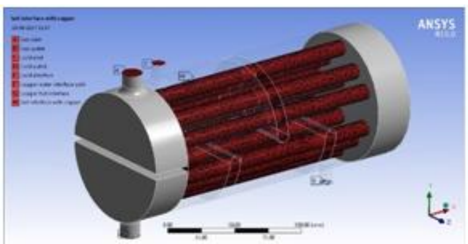

Figure 12 Heat exchanger showing hot interface with copper tubes.

In the above Figure 12 shows that copper material is used in tubes. Inside the tube is hot water is flowing and cold water is flowing inside the shell means outer surface of the copper tube. There are 12 tubes are used in the total experimental setup, which is arranged in triangular pitch like zigzag manner. Baffles are used to support the tubes and direction of fluid flow. Total three baffles are used in this setup. Copper material is used in place of mild steel because thermal conductivity of copper is more as compare to mild steel since higher thermal conductivity means better heat transfer rate.

3.5 RESULTS AND DISCUSSION:

Under the Above boundary condition and solution initialize condition simulation was set for 50-50 iteration. Convergence of Simulation: The convergence of Simulation is required to get the parameters of the shell and tube heat exchanger in outlet. It also gives accurate value of parameters for the requirement of heat transfer rate. Continuity, X-velocity, Y-velocity, Z-velocity, energy, k, epsilons are the part of scaled residual which have to converge in a specific region. For the continuity, X-velocity Y-velocity, Z-velocity, k, epsilons should be less than 10-4 and the energy should be less than 10-7.If the seall values in same manner then solution will be converged.

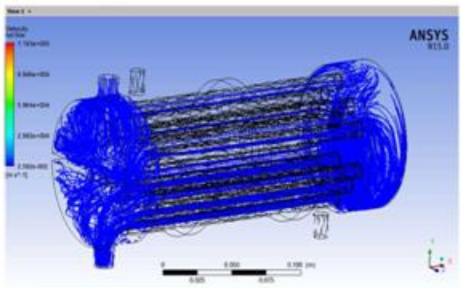

Figure 13 CFD Velocity profile of heat exchanger.

In the MODEL tree option the energy equation should be turned on for calculating the temperature profile for the heat exchanger. For turbulent model standard K-ε model was used. For this viscous model standard k-ε model was turned on.

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

In the above figure shows the flow of fluid inside the tube and inside the shell. From CFD analysis we flow the both the fluid from tube where hot fluid is flowing and from shell side cold fluid is flowing. In the analysis part shows the velocity profile of fluid flow in shell and tube type of heat exchanger. In the CFD analysis the fluid flow is turbulent flow. For selecting the material we have to go for problem setup tree and then go for the MATERIAL option then go for change and edit option where we can select the material from fluent database.

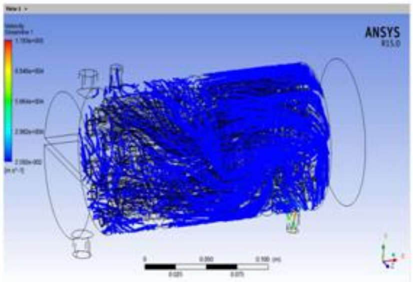

Figure 14 CFD velocity streamline profile of heat exchanger

After selecting the suitable material (copper for solid and water for fluid), we can change theirproperty according to our requirements. Then we have to specify the domain or the cell zonecondition. For inner and outer fluid it should be water-fluid and for solid it should be copper.

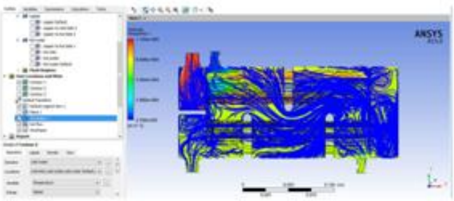

Figure 15 CFD velocity profile showing flow of hot and cold fluid.

In the above figure shows that streamline1 of hot fluid and cold fluid. The different color shows the maximum and minimum velocities of fluids. For setting the boundary condition go to BOUNDARY CONDITION option in problem set up tree then chose the different boundary condition zone with specifying the type of boundary (velocity inlet/pressure outlet/wall) etc.

Then in each boundary condition go for EDIT option then specify the values according to the requirements.

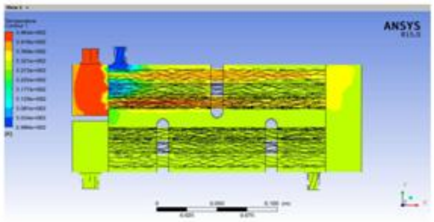

Figure 16 Temperature profile of heat exchanger of hot and cold fluid

In the above figure shows the variation of minimum and maximum temperature shows. The maximum temperature found at hot water inlet and minimum is found at cold water inlet. Herewith temperature contour showing edges of the tube to make the body part transparent with the help of CFD analysis. Here in the upper part of the tubes is showing near the maximum temperature because here is passing start and contact with cold water surface, So because of transfer of heat from high to low temperature is decreasing continuously of hot water.

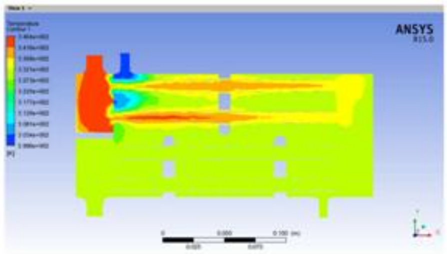

Figure 17 Temperature contour profile of heat exchanger of hot and cold fluid

In the above analysis of temperature profile shows hot fluid without any edges of the tubes which is similar as Figure 16.

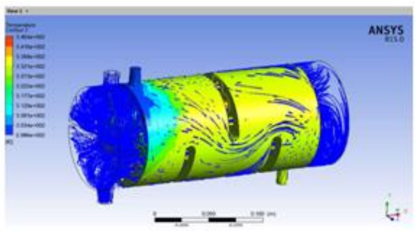

Figure 18 CFD model of temperature contour showing cold fluid flow

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

In the above diagram it shows only the variation of temperature contour of cold fluid. The minimum temperature is found at inlet and average temperature is found at middle part of the setup which shows in yellowish color.

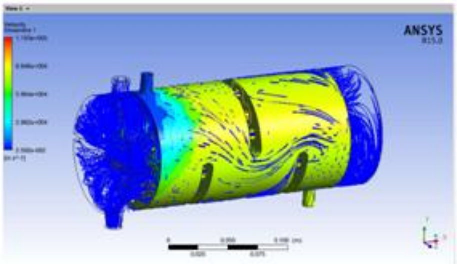

Figure 19 velocity streamline profile of fluid flow inside heat exchanger.

In the above Figure 19 shows the velocity streamline profile of cold fluid. In The whole setup the cold water is flowing through the shell which is passes by the number of baffles used. The flow of cold fluid is turbulent flow because flow of fluid is non-uniform.

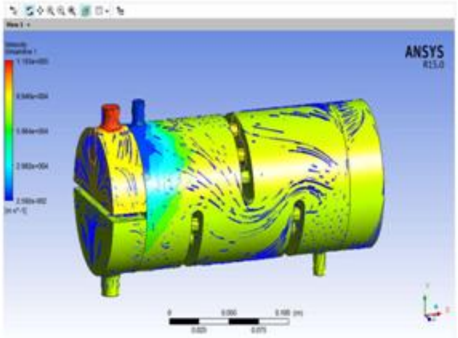

Figure 20 CFD velocity streamline of heat exchanger.

In the above figure velocity streamline of both the fluids both hot and cold fluids passes through the tubes and shell respectively It shows the turbulence of the fluid flow in nearer to the baffles the fluid is showing more turbulence because of it passes through the baffles. In the above figure it shows the maximum and minimum values of the fluid flow.

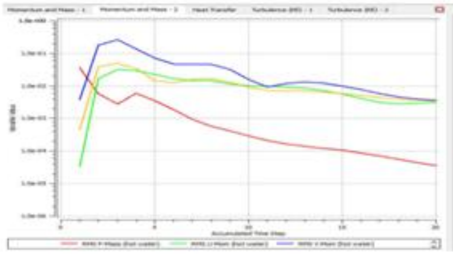

Figure 21 Graph shows that mass momentum of hot water.

In the above Figure 21 shows that mass and momentum of hot water. The graph plotted by CFX solver using the boundary conditions. Red line shows the mass flow rate of hot water and remaining are the momentum of the hot fluid

TABLE 4: TABULATION FOR DIFFERENT MASS FLOW RATE

CI	CO	HI	HO	TIME/COLD	TIME/HOT	Mass flow rate kg/s	Mass flow rate kg/s	Heat rejected/aided (W)	
						COLD WATER	HOT WATER	Hot fluid	Cold fluid
298.6	331.52	346.21	331.58	41.04	38.37	0.012183	0.013031	807.8	1675.64
298.6	340.15	356.4	335.21	41.04	38.37	0.014183	0.015031	1352.21	1945.25
298.6	345.56	366.25	333.11	41.04	38.37	0.016183	0.017031	1897.02	2361.56
298.6	351.00	376.96	329.99	41.04	38.37	0.018183	0.019031	2443.14	2721.66
298.6	360.12	386.45	335.07	41.04	38.37	0.020183	0.021031	2988.16	3125.29

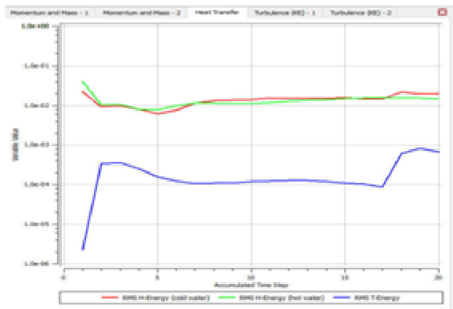

Figure 22 Graph shows heat transfer rate of hot and cold fluids.

In the above Figure 22, it shows the heat transfer rate for hot and cold fluid. Red line shows the heat transfer rate for cold fluid, green one show for the hot fluid and blue for total heat transfer rate. The graph can be plotted by CFX solver using least algorithm method.

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

We can find from the graph the energy absorbed by the cold water and heat release by the hot water is approximately same. The error is within 3% so there shows a little bit variation which is very close to the experimental values.

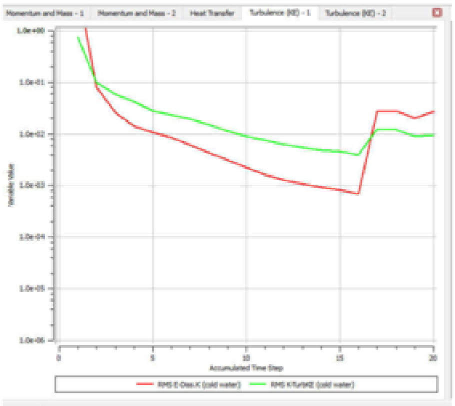

Figure 23 Graph shows the turbulence kinetic energy for cold water.

In the above figure shows the turbulence kinetic energy for the cold water. The turbulence kinetic energy of cold water is more than the hot water. Tube side to high pressure fluids (Shell side high pressure increases the thickness of shell and can have great implications on cost of the exchanger). Remember, shell cost forms the major cost of the exchanger in most cases. Shell side to viscous fluids as turbulence, which can be more easily induced on shell side might be difficult(note viscous and fouling fluids are not essentially the same thing and are often confused as high viscosity implying high fouling). And then there are tube inserts which can give you that effect on tube-side as well.

CONCLUSION:

In this shell and tube type heat exchanger, we have been done experimental and CFD analysis. In experimental analysis shell and tube type heat exchanger we are used 12 tubes, three baffles and water as a working fluid. And finally we will find the cold outlet, hot outlet and heat transfer. We design the same type heat exchanger by using ANSYS 15.0 software.

Computational Fluid Dynamics is done according to experimental setup. The shell and tube type heat exchanger have been designed and analyzed in the C.F.D. software. Then we got the hot outlet, cold outlet and heat transfer values is very nearer to experimental data i.e. very less error (within 3%). After we done analysis based on our calculated values for stream line, temperature contour and velocity analysis for both hot and cold fluid. CFX have been used to plot the graph for momentum and mass, turbulence flow for both hot and cold fluid and also shows the heat transfer will be effective with same experimental data.

FUTURE SCOPE OF WORK:

In future, the work can be extended for analyzing heat exchanger with presence of fin and more baffles. Analysis can be focused by varying Fluid thermal properties and Shell and Tube materials and its thickness and dimensions.

REFERENCES:

[1] Usman Ur Rehman “Heat Transfer Optimization of Shell-and Tube Heat Exchanger throughCFD Studies”, Goteborg, Sweden 2011, Master’s Thesis 2011:09, Chalmers University ofTechnology.

[2] Patil Vinay “Validation of Plate Heat Exchanger Design Using Cfd”, ISSN 2278 – 0149Vol. 2, No. 4, October 2013.

[3] Sunil B. Revagade (2015) “Analysis of different type of tubes to optimize the efficiency ofHeat Exchanger”.

[4] Vindhya Vasiny Prasad Dubey “Performance Analysis of Shell & Tube Type HeatExchanger under the Effect of Varied Operating Conditions”, May- Jun 2014, IOSR-JMCE.

[5] D.Bhanuchandrarao “CFD Analysis And Performance Of Parallel And Counter Flow

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

InConcentric Tube Heat Exchangers" Vol. 2 Issue 11, November – 2013, ISSN: 2278-0181

[6] Yunus A. Cengel "Heat transfer A Practical Approach second edition pp 667 to 700"

[7] B.Jayachandriah (2014) "Thermal Analysis of Tubular Heat Exchangers Using ANSYS"ISSN:(2319-6890) 22nd March 2014

[8] Vinodkumar "Improvement of Heat Transfer Coefficients in a Shell and Helical Tube HeatExchanger Using Water/Al2o3 Nanofluid", June-2015, IRJET.

[9] HetalKotwal "CFD Analysis of Shell and Tube Heat Exchanger-A Review",March2013,IJESIT.

[10] Amol S. Niphade" Design, CFD Analysis of Shell & Tube Heat Exchanger with CFDAnalysis for Dairy Application", Volume 2, Issue 9 , March 2016, IJAFRSE.

Author's Details:

Pappala Santosh Saikumar
M.Tech.[Thermal Engineering] Student, Department of Mechanical Engineering, Sri Vaishnavi College of Engineering, Srikakulam, Andhra Pradesh 532001, India.

Sattaru Suresh Babu
Assistant Professor Mr. Sattaru Suresh Babu Was Born In Andhra Pradesh, India. He Has Received M.E. [HTES] From A.U. AP, India. He Is Working As Assistant Professor In Mechanical Engineering Dept, Sri Vaishnavi College Of Engineering, Srikakulam, India.

Kotipatruni Ratnakara Rao
Assistant Professor, Was Born In Andhra Pradesh, India. He Has Received M.Tech. [Machine Design] From JNTU, Kakinada. AP, India. He Is Working As Assistant Professor In Mechanical Engineering Dept, Sri Vaishnavi College Of Engineering Srikakulam, India.