International Journal of Scientific & Engineering Research Volume 12, Issue 2, February-2021
ISSN 2229-5518

Design Fabrication and Performance analysis of a Shell and tube heat exchanger using Computational fluid dynamics

Shishir D Athreya, Dharshan B G, Punith S M, Prabhava M, Prabhu V,

Abstract — In this study, endeavors to design a Shell and tube heat exchanger by modeling in Solidworks software, which is having a tube of inner diameter 7.75mm and outer diameter of 9.52mm, five tubes of length 400mm were designed. The design considerations were made by referring to TEMA Standards. The analysis of this model was carried out by using fluent package of ANSYS®. Water was used as a medium of heat transfer. The model was analyzed for same and different mass flow rates at the inlets of cold and hot fluids and from the simulations results it was found that the heat transfer rate increased for different flow rates at the inlets of cold and hot fluids. The prototype was fabricated and tested. The results from experimentation was compared with the results of computational method to validate the performance.

Keywords - ANSYS fluent, Mass flow rate, Shell and tube heat exchanger, TEMA Standard, Temperature difference

1 INTRODUCTION

The application of the principles of heat transfer to the design of equipment to accomplish a certain engineering objective is of extreme importance, for in applying the principles to design, the individual is working toward the important goal of product development for economic gain [8].

Shell and Tube Heat exchanger is a classification based on construction and it is further classified as Crossflow to tubes and Parallel flow to tubes.

A heat exchanger is a device which used to transfer heat from one fluid to another through a solid medium or interface. From the results and data obtained from CFD it was concluded that the $K-\epsilon$ turbulence model provided better suitability. The increasing effectiveness of the heat exchanger increases its performance in its respective application [1].

The efficiency of the shell and tube heat exchanger depends on different factors like tube diameter, tube length, number of tubes, number of baffles, mass flow rate of the fluid, temperature of the fluid flowing [2].

It was clear that with the increase in the shell side area, the mass flow rate of cold fluid increased there by enhancing heat transfer rate [3][5].

The use of baffles was a remarkably effective role for appropriate development of flow of fluid and temperature's profiles. It helps in distribution of flow and temperature. According to the results, although increase of entry cross area helps to make better distribution of fluids, but large scale of it may increase pressure drop and operation expenses [4].

In this paper our objective is to design a shell and tube heat exchanger and fabricate it according to TEMA standards.

- *Author name is Shishir D Athreya, Student, Department of Mechanical Engineering, Don Bosco Institute of Technology, Bangalore, India. PH-+91-8095198275. E-mail: shishir982@gmail.com*
- *Co-Author name is Dharshan B G, Associate Professor, Department of Mechanical Engineering, Don Bosco Institute of Technology, Bangalore, India. PH-+91-9880601016. E-mail: dharshubg@gmail.com*

The performance analysis of shell and tube heat exchanger can be determined by changing tube dimensions, geometry, mass flow rate of fluid, length of fluid travel and number of baffle plates. The overall heat transfer coefficient will be determined for the same mass flow rate and different mass flow rates. Experimental results are compared with the numerical analysis carried out in ANSYS 17.0 solver by using fluent package. This paper is organized as follows. In section 2, the design calculations are made for certain parts of shell and tube heat exchanger using TEMA standards. Section 3 contributes to study of fluid flow analysis of the model. In section 4 and 5, the details about the fabrication of experimental set up and experimentation is explained briefly. The results of the analysis and experimentation are compared in section 6. Finally, conclusions are summarized in section 7.

2 SHELL AND TUBE HEAT EXCHANGER DESIGN PROCEDURE AND CALCULATIONS

The 3D modeling of the heat exchanger is done in Solid works 2017 software as per the calculation data. The entire assembly consists of 11 parts and the detailed design procedure for few of the parts are mentioned below.

2.1 Tube design and selection

From the "Section-9 Table D-7m" (TEMA Standard) [6] a tube of outer and inner diameter 9.52mm, 7.75mm was selected. U tube is selected for the heat exchanger and the material of the tube is Copper. U-tube requirement as per TEMA is selected from section RCB-2.31[6].

According to TEMA, section RCB-2.31

$$t_0 = t_1\left[1+\frac{d_0}{4R_b}\right] \tag{1}$$

Where t_0= initial tube thickness prior to bending in mm.

t_1= minimum tube wall thickness calculated by code rules for a straight tube subjected to same pressure and metal temperature in mm.

d_0= OD of tube in mm.

R_b= mean bend radius in mm.

For copper = 4-8 % (thinning for full expansion)

Considering $8\% = 0.08 \times 0.889 = 0.07112$
$t_1 = t_0 - 8\%\, t_0 = 0.889 - 0.07112$
$t_1 = 0.81788\text{mm}$
$0.889 = 0.81788\left[1 + \left(\frac{9.52}{4\times R_b}\right)\right]$
$R_b = 27.35\text{mm}.$

The number of tube passes were designed by referring to Fig.1.and Fig.2. shows the 3D model of the U-tube.

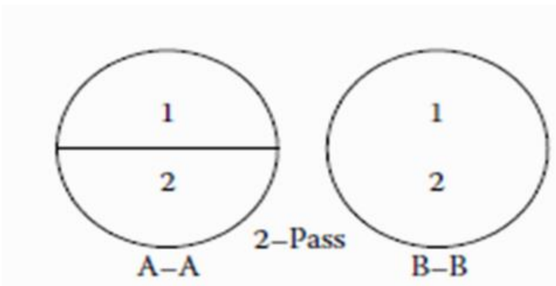

Fig.1. Typical tube side partitions for U-tube. Note-A–A: Front view and B–B: Rear view

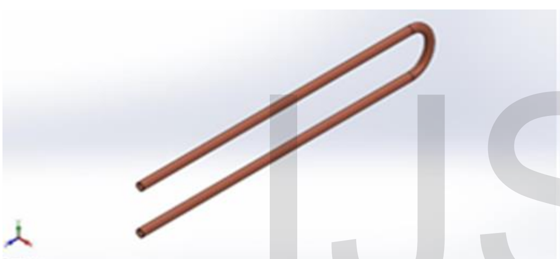

Fig.2. 3D Model of U-tube

Tube spacing is done as per the TEMA Standards table RCB-7.22M [6]. From the table, the pitch (p) for the tube of outer diameter 9.52mm was 12.70mm.Fig.3. shows the 3D model of tubes spacing.

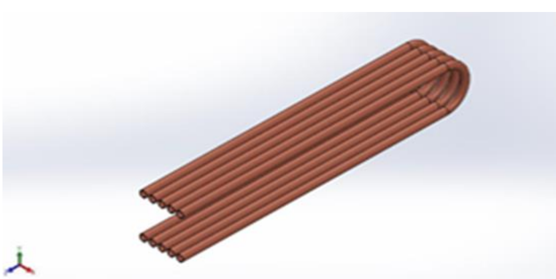

Fig.3. Illustration of tube spacing

2.2 Tube Sheet design calculation

Total thickness

$$t_s = \frac{FG_p}{3}\sqrt{\frac{P}{K\times f}} \qquad (2)$$

Where $F = 1.25$ for U- Tube Sheet
$G_P =$ Shell inner diameter $= D_s = 109.3\text{mm}$
$f = Allowable\ stress = 41.368N/mm2$

$K = \left(1 - \frac{0.785}{\left(\frac{PT}{d_0}\right)^2}\right)$ for Square pitch
$(K = 0.5)$ for Square rectangular pitch
$t_s = (1.25 \times 109.3/3) \times \sqrt{(2.09/0.5 \times 41.368)}\,)$
$t_s = 14.476mm.$

Mild steel was used as the tube sheet material.Fig.4. shows the geometric model of tube sheet.

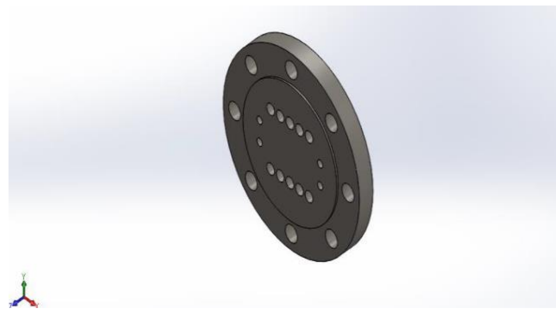

Fig.4. Geometric model of tube sheet

Segmental baffles were used in this Shell and tube heat exchanger. Baffle thickness is selected based on the shell inner diameter. From table (B-4.41) [6] the thickness was found to be 1. 6mm. Distance of 50mm was maintained between each baffle. Spacers of suitable length were used to maintain this dimension. Fig.5. shows the arrangement of baffles.

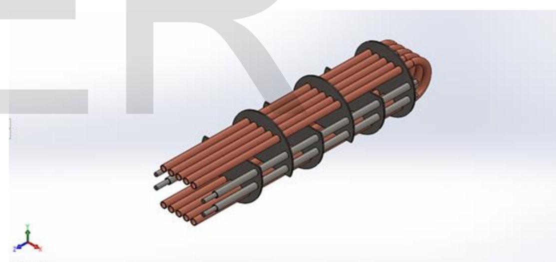

Fig.5. Illustration of baffle spacing

The front and rear ends of the shell and tube heat exchanger are selected form heat exchanger nomenclature, and it is of B-type and U-type. Fig.6. shows the exploded view of the assembly. Gaskets are used to avoid leakages in shell and tube heat exchanger. Asbestos was used for the gaskets.

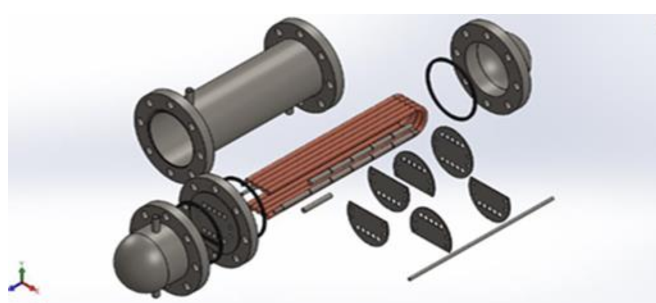

Fig.6. Exploded view of the assembly

2.3 Material properties

Copper was selected as the tube material; asbestos was used for the gasket material and for the rest of the heat exchanger mild steel was used as the material. The properties for these materials as mentioned in the Table1.

Table 1. Material properties

Material	Thermal conductivity$\left(\frac{W}{m-k}\right)$	Density$\left(\frac{kg}{m^3}\right)$
Copper	386	8940
Mild steel	27.0	7850
Asbestos	0.08	285

3. FLUID FLOW ANALYSIS

The computational model with 5 segmental and one full baffle is shown in Fig.7., the simulated model had six cycles of baffles in the shell side direction and the total number of tubes is 5. The inner side of the shell bound the whole computation domain and everything in the shell contained in the domain. The inlet and outlet of the domain relate to the corresponding tubes.

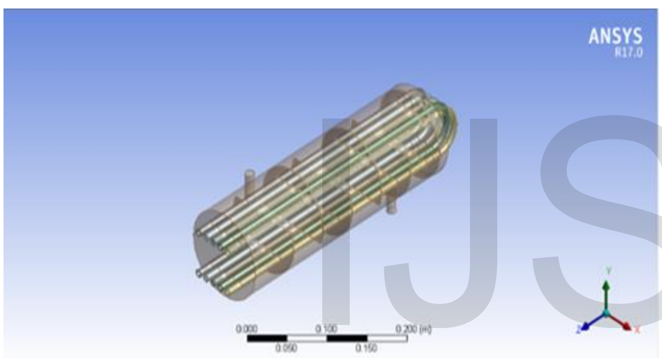

Fig.7. Geometry of the model imported for analysis

Initially, a relatively coarser mesh was generated with 1.6 Million elements. This mesh contains hexahedral elements having triangular at the boundaries for shell and hexahedral elements having quadrilateral faces were used for the tube. Structured hexahedral elements was used. To decrease the numerical diffusion fine mesh was used particularly near the wall region.

ANSYS® FLUENT® v17.0 was used for the simulation of this model. In the Fluent setup pressure based type solver was selected, absolute velocity formation, transient time was selected and gravitational acceleration of $-9.81m/s^2$ was selected in y direction for the simulation of the model imported. In the setup tree model option, energy calculation was on and the viscous was set as realizable $K-\epsilon$, standard wall function. Boundary condition were selected for the inlets, outlets of both hot and cold fluids and mesh interface was created. In first case inlets of both fluids were same. Mass flow rate of $0.017kg/s$ was used. In the second case at the inlet of hot fluid a mass flow rate of $0.017kg/s$ and temperature of $333K$ was assigned. Back flow temperature of $300K$ was set at the hot fluid outlet. Across each tube, a mass flow rate of $0.034kg/s$ and $300K$ temperature was assigned.

Boundary condition were selected for the walls of the U tube after the selection of inlets and outlets flow conditions and mesh interfaces are created. In the reference value area set as $1m^2$, density of 1000 kg/m^3, enthalpy of 229485 J/kg, length of $1m$, temperature of $333K$ and specific heat of 1.4 was considered.

Pressure velocity coupling with coupled scheme was selected for solution methods, the value of pressure was 0.7, density 1, body force 1, momentum 0.2, turbulent kinetic and turbulent dissipation rate was set at 1, energy and turbulent viscosity was 1(default conditions). Solution initialization was hybrid method and solutions were initialized from inlet with $300K$ temperature.

3.1 RESULTS OF CFD ANALYSIS

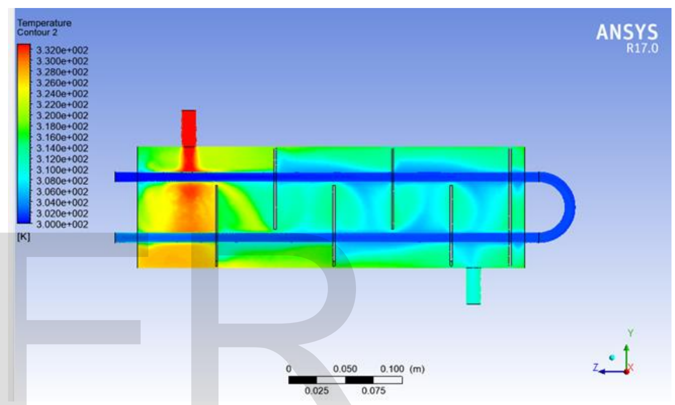

Fig.8.Temperature variation in the heat exchanger after 250 iterations

The Shell and heat exchanger was analyzed for two different mass flow rates.

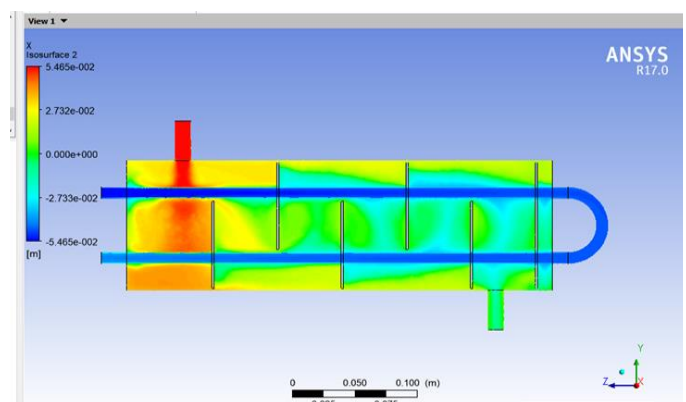

Fig.9. Temperature variation in the heat exchanger after 5500 iterations

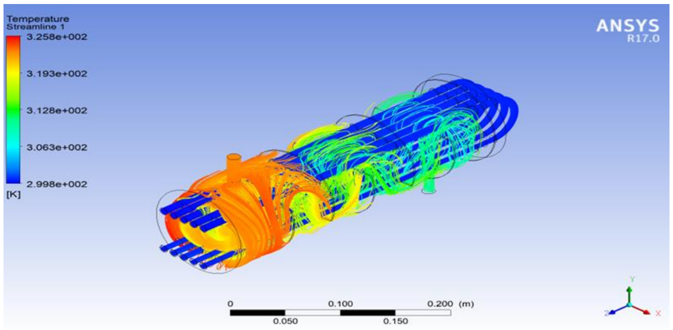

Fig.10.Fluid flow representation in the form of streamlines

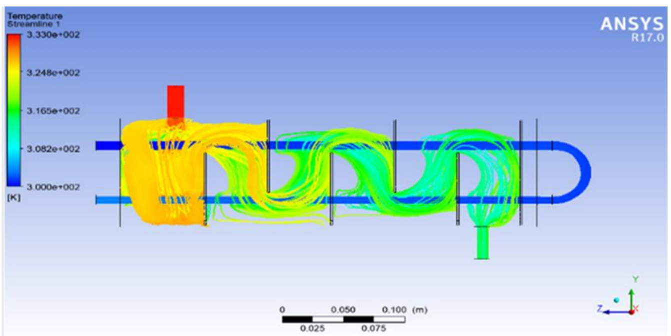

Fig.11.Front view of fluid flow representation in the form of streamlines

Fig.8. shows the temperature contour after 250^{th} iteration.Fig.9., Fig.10. and Fig.11. shows the change in temperature as the fluid flows through the cross-section.

Two different mass flow rates were considered during the analysis and the results of this analysis are given in the Table 2.

Table 2. Results of the temperature difference at the outlet of the heat exchanger due to different mass flow rates.

Mass flow rate at cold water inlet (kg/sec)	Mass flow rate at hot water inlet (kg/sec)	Outlet Temperature of Shell side (K)	Outlet Temperature of Tube side (K)
0.034	0.017	323	315
0.017	0.017	325	308

4. FABRICATION

Fabrication of the model was done in the following manner: Initially, a Standard 4-inch pipe was cut according to the shell dimensions i.e. (340 mm), material of the pipe is mild steel. Copper tubes were selected according to the design i.e. (OD-9.52mm ID-7.75mm TEMA specification). These tubes were bent according to the mean radius calculated in the calculation part. Standard 4-inch dish ends were selected and are welded to the plain flange using the TIG welding process as shown in Fig.12 and holes are drilled for the nozzles as per design specifications.

Fig.12. TIG welding

Plain flanges were welded to shell. Baffle as shown in Fig.13. was cut according to the dimensions mentioned in the draft using laser-cutting process.

Fig.13. Baffle

Spacers were cut as per the specifications. Fig.14. shows assembly of tubes, baffles, and the spacers. Holes are drilled for the blind flange as per tube sheet specifications and tie rods are cut according to the specifications. Fig.15. shows saddles welded to the shell for support. Nozzles are welded to the shell and the bonnet end.

Fig.14. Tube bundle assembly

Fig.15. Shell welded with saddle and flanges

The inner surface of the shell and the bonnet ends are coated to prevent corrosion. Once the assembly of entire shell and tube heat exchanger is done materials required for, pipe connections are arranged or selected as per requirement and the flow meters are connected to the elbows and valves. Final connections are made for testing of the apparatus. The apparatus was tested to check whether there are any leakages in the assembly or setup.

Fig.16. Experimental setup

5. EXPERIMENTATION

The apparatus is tested again for different mass flow rates (m) and the overall heat transfer coefficient (U) is determined with the help of inlet and outlet Temperature readings obtained during testing. Reynolds number (R_e), Nusselt number (N_u), co-efficient of convective heat transfer (h) are calculated.

Initially the testing is done for equal mass flow rates i.e. The inlet mass flow rate of cold fluid is equal to the inlet mass flow rate of hot fluid. The readings of this are tabulated and the calculations are done accordingly. Later the mass flow rate of cold fluid is doubled, the readings are tabulated, and calculations are done accordingly. The results of same mass flow rates are given in Table 3.

Table 3. Results for same mass flow rates

Mass flow rate of hot water in (LPM)	Hot water inlet temperature (in ℃)	Hot water outlet temperature (in ℃)	Mass flow rate of cold water in (LPM)	Cold water inlet temperature (in ℃)	Cold water outlet temperature (in ℃)
1	48	46.4	1	30	31.8

Correction Factor

$R = \frac{(T_1-T_2)}{(t_2-t_1)} = \frac{(321-319.4)}{(304.8-303)} = 0.888$

$P = \frac{(t_2-t_1)}{(T_1-t_1)} = \frac{(304.8-303)}{(321-303)} = 0.1$

LMTD Counterflow

$LMTD = \frac{[(T_1-t_2)-(T_2-t_1)]}{\ln\left[\frac{(T_1-t_2)}{(T_2-t_1)}\right]}$

$LMTD = \frac{[(321-304.8)-(319.4-303)]}{\ln\left[\frac{(321-304.8)}{(319.4-303)}\right]} = 15.384$

$LMTD = 15.384$

$EMTD = Correction\ factor \times LMTD = 0.95 \times 15.384$

$EMTD = 14.614$

$M_h \times C_h = 0.017 \times 4187$ $(Since\ 1LPM = 0.017kg/sec)$

$C_{min} = 71.179\ J/K-sec$

$M_c \times C_c = 0.017 \times 4187$

$C_{max} = 71.179\ J/K-sec$

$\varepsilon = 2[1 + C\ (1 + C^2)^{0.5}) \times 1 + exp\ [-N\ (1 + C^2)^{0.5}]/1 - exp\ [-N\ (1 + C^2)^{0.5}]]^{-1}$

$\varepsilon = 0.092$

$Q = \varepsilon \times C_{min}\ (T_1 - t_1) = 0.092 \times 71.179(321 - 303)$

$Q = 117.872\ J/sec$

$R_{e_i} = \left(V_i \times \frac{d_i}{u}\right) = \left(\frac{4m}{\pi d_i u}\right) = \frac{(4\times0.017)}{(\pi\times7.75\times10{-}3\times855\times10^{-6})}$

$R_{e_i} = 3266.564$ greater than 2300

Hence, flow is turbulent

$N_{u_i} = 0.023(R_{e_i})^{0.8} \times (P_r)^{0.4} = 0.023 \times (3266.564)^{0.8} \times (5.84)^{0.4}$

$N_{u_i} = 30.169$

$N_{u_i} = \frac{(h_i \times d_i)}{K}$

$30.169 = \frac{(h_i\times7.75\times10^{-3})}{0.613}$

$h_i = 2386.27\ W/m^2K$

$R_{i_o} = \left(\frac{4m}{\pi d_0 u}\right) = \frac{(4\times0.017)}{\pi\times9.52\times10^{-3}\times855\times10^{-6}}$

$R_{i_o} = 2659.23$ greater than 2300

Hence, flow is turbulent

$N_{u_0} = 0.023(R_{i_o})^{0.8} \times (Pr)^{0.4} = 0.023 \times (2659.23)^{0.8} \times (5.84)^{0.4}$

$N_{u_0} = 25.591$

$N_{u_0} = \frac{(h_0 \times d_0)}{K}$

$25.591 = (h_0 \times 9.52 \times 10^{-3})/0.639$

$h_0 = 1717.715\ W/m^2K$

$\frac{1}{U} = \frac{1}{h_i} + \frac{1}{h_0} = \frac{1}{2386.27} + \frac{1}{1717.41} = 1.00 \times 10^{-3}$

$U = 1000 \frac{W}{m^2K}$

The results of different mass flow rates are given in Table 4. Calculations for the same were made and the steps are similar to the above calculations.

Table 4. Results for different mass flow rates

Mass flow rate of hot water in (LPM)	Hot water inlet temperature (in ℃)	Hot water outlet temperature (in ℃)	Mass flow rate of cold water in (LPM)	Cold water inlet temperature (in ℃)	Cold water outlet temperature (in ℃)
1	51.6	45.3	2	28.4	31.4

Correction Factor

$R = \frac{(T_1 - T_2)}{(t_2 - t_1)} = \frac{(324.6 - 318.3)}{(304.4 - 301.4)}$

$R = 2.1$

$P = \frac{(t_2 - t_1)}{(T_1 - t_1)} = \frac{(304.4 - 301.4)}{(314.6 - 301.4)}$

$P = 0.129$

LMTD Counterflow

$LMTD = \frac{[(T_1 - t_2) - (T_2 - t_1)]}{\ln\left[\frac{(T_1 - t_2)}{(T_2 - t_1)}\right]} = \frac{(324.6 - 304.4) - (318.3 - 301.4)}{ln\,[324.6 - 304.4/\,318.3 - 301.4]}$

$LMTD = 18.5$

$EMTD = Correction\ factor \times LMTD = 0.95 \times 18.5$

$EMTD = 17.575$

$M_h \times C_h = 0.017 \times 4187\ (Since\ 1LPM = 0.017kg/sec)$

$C_{min} = 71.179\ J/K - sec$

$M_c \times C_c = 0.034 \times 4187$

$C_{max} = 142.35\ J/K - sec$

$\varepsilon = 2[1 + C\ (1 + C^2)^{0.5})\ \times 1 + exp\ [-N\ (1 + C^2)^{0.5}]/1 - exp\ [-N\ (1 + C^2)^{0.5}]]^{-1}$

$\varepsilon\ = 0.092$

$Q =\ \varepsilon\ \times C_{min}\ (T_1 - t_1) = 0.092 \times 71.179(321 - 303)$

$Q = 151.92J/sec$

$R_{e_i} = \left(V_i \times \frac{d_i}{u}\right) = \left(\frac{4m}{\pi d_i u}\right) = \frac{(4 \times 0.034)}{(\pi \times 7.75 \times 10^{-3} \times 855 \times 10^{-6})}$

$R_{e_i} = 6533.128 \qquad greater\ than\ 2300$

Hence, flow is turbulent

$N_{u_i} = 0.023(R_{e_i})^{0.8}\ \times\ (P_r)^{\,0.4}\ = 0.023 \times\ (6533.128)^{0.8} \times (5.84)^{0.4}$

$N_{u_i} = 52.491$

$N_{u_i} = \frac{(h_i \times d_i)}{K}$

$52.491 = \frac{(h_i \times 7.75 \times 10^{-3})}{0.613}$

$h_i\ = 4151.78W/m^2K$

$R_{i_o} = \left(\frac{4m}{\pi d_0 u}\right) = \frac{(4 \times 0.017)}{\pi \times 9.52 \times 10^{-3} \times 855 \times 10^{-6}}$

$R_{i_o} = 2659.23 \quad greater\ than\ 2300$

Hence, flow is turbulent

$N_{u_0} = 0.023(R_{i_o})^{\,0.8} \times\ (Pr)^{\,0.4}\ =\ 0.023 \times\ (2659.23)^{0.8} \times (5.84)^{0.4}$

$N_{u_0} = 25.591$

$N_{u_0} = \frac{(h_0 \times d_0)}{K}$

$25.591 =\ (h_0 \times 9.52 \times 10^{-3})/0.639$

$h_0 = 1716.5W/m^2K$

$\frac{1}{U} = \frac{1}{h_i} + \frac{1}{h_0} = \frac{1}{4151.78} + \frac{1}{1716.5}$

$1/U = 1.00 \times 10^{-3}$

$U = 1214.41\ W/m^2K$

6. RESULTS AND DISCUSSION

The performance analysis of the shell and tube heat exchanger was carried out experimentally and the with suitable computational methods. The focus was mainly on the temperature of the cold fluid outlet. The analysis carried out by using same and different mass flow rates of the fluids at the inlet valves of both cold and hot fluid. The comparison between the fluid flow analysis and the experimentation can be summarized as follows:(1) The temperature difference is more in case different mass flow rates when compared to the same mass flow rates. Nearly 5℃ difference was obtained at the outlet of the cold fluid from the experimentation and 8℃ difference was obtained from the CFD analysis at the outlet of the cold fluid.

(2) From the calculations it was found that the overall heat transfer coefficient (U) was more for the different mass flow rates at inlet valves when compared to the one with the same flow rate at the inlets.

7. CONCLUSION

The heat transfer for the model tested with same and different flow rates. Initially the model was tested and analyzed for a mass flow rate of $0.017kg/s$ at both the inlets. The temperature at the outlet of the cold fluid was $308K$ from the CFD analysis and from the experimentation it was found to be $304.8K$. In the second case the mass flow rate was increased at the cold fluid inlet. A mass flow rate of $0.034kg/s$ was used. Temperature at the outlet of the cold fluid was increased to $315K$ in CFD analysis and $304.4K$ in experimentation and it is to be noted that the inlet temperature of cold fluid was about $301.4K$ in second case. The different mass flow rates helped to get better temperature difference between inlet and outlet. The heat transfer can be further increased by using helical baffles at different angles instead of segmental baffles.

ACKNOWLEDGMENT

The authors wish to thank Mr. Raghavendra and Mr. Vinod of HBM Heat Ex LPP for their valuable guidance and support.

REFERENCES

[1] Avinash P. Shinde, Mahadev L. Shinde, Sagar S. Yadav, Manasvi R. Surve, A. S. Futane, Mainak Bhaumik "Performance Analysis of Heat Exchanger using CFD" International Journal of Innovative Research in Science, Engineering and Technology ISSN(Online): 2319-8753 ISSN (Print): 2347-6710.

[2] K. Maheswari Devi, G.V. Nagamani "Design and Thermal Analysis of Shell and Tube Heat Exchanger by Using Fluent Tool" International Journal & Magazine of Engineering, Technology, Management and Research ISSN No: 2348-4845.

[3] Mr. Santosh K Katarki, Mr. Anandkumar S Malipatil "CFD Analysis of Shell and Tube Heat Exchanger for Heat Transfer Capabilities" International Journal of Engineering and Techniques ISSN: 2395-1303.

[4] Pooja J. Pawar, Experimental investigation of shell-and tube heat exchanger with different type of baffles, International Journal of Current Engineering and Technology Accepted 15 June 2016, Available online 20 June 2016, Special Issue-5 (June 2016).

[5] Nakka Sita Rama Raju, V Jaya Prasad, S Kamaluddin and A Sunny Kumar. Thermal Analysis of Shell and Tube Type Heat Exchanger to Demonstrate the Heat Transfer Capabilities of Various Thermal Materials Using Ansys. *International Journal of Mechanical Engineering and Technology*, 8(2), 2017, pp. 160–166.

[6] TEMA Standards

[7] Kuppan Thulukkanam, Heat Exchanger Design-Handbook, Dekker Mechanical Engineering, 2013.

[8] J.P. HOLMAN, Heat Transfer, Tata Mc-Graw Hill international publishers 9th edition, 2002.

[9] Anil Kumar Samal Roll, "Shell and tube heat exchanger design using CFD tools", B-tech thesis, Department of Chemical Engineering, NIT Rourkela,769008.