Thermal performance of a Divided-flow type Shell and Tube Compact Heat Exchanger

Dr. Nagaraj S[1], Abhijeeth Nagaraj*[2] and Karthik N[2]

[1] Department of Civil Engineering, MVJ College of Engineering, Bangalore, India.
[2] Department of Mechanical Engineering, School of Engineering and Technology, Jain (Deemed-to-be University), Bangalore, India

*n.abhijeet@jainuniversity.ac.in

Abstract. Shell and Tube heat exchanger is widely used as heat-transfer equipment in various process industries. The paper attempts to understand the flow, heat transfer characteristics and the effectiveness of the divided-flow type heat exchanger. The study is based on the CFD based simulation methods and the experimental methods. The study provides the flow and heat-transfer characteristics of the divided-flow type heat exchanger by using validated Computational Fluid Dynamics (CFD) model. Results are obtained for the temperature, velocity and turbulence intensity distributions in the flow domain. The results from the simulation and experimental study are used in determining the effectiveness (ξ) of the heat exchanger using Logarithmic Mean Temperature Difference (LMTD) and Number of Transfer Units (NTU). The simulation and experimental performance are found to be in good agreement highlighting the minimization of losses in the experimental setup. This confirms the accuracy of the flow-based heat and turbulence distributions. The effectiveness value shows that the heat exchanger can be used for compact applications

Keywords: CFD, Divided flow, Effectiveness, Heat Exchanger, LMTD, Shell and Tube.

1. Introduction

Heat exchangers are one of the critical components in thermal plants, where the heat energy needs to be transferred from one fluid to another. The device is a closed system where the heat energy is allowed to transfer but the mixing of the fluids physically is restricted. Heat exchangers are available in variety of designs. The common heat exchanger principle follows a parallel or counter flow setup for transfer of heat. However, both of the setups have a certain limitation during the heat transfer mechanism.

The shell and tube heat exchangers are widely used in thermal and chemical process plants. The common operational setup for a shell and tube type of heat exchanger is parallel flow or counter flow. However cross flow setup is observed to provide a compact design for heat exchanger. The flow characteristics of the parallel, counter and cross flow heat exchanger can be amalgamated in divided flow type heat exchanger, which behaves as a parallel flow, counter flow and cross flow heat exchanger. Studies have been performed on improving the heat exchanger using Nano Fluids using counter flow plate type heat exchanger by Fard[1]. The study on similar nano fluids utilize a laminar flow-based shell and tube parallel type heat exchanger by Akhtari[2] show an improvement in the heat transfer capacity. However, studies show that the improvement in the flow direction plays a vital role in heat transfer. Hence need of an effective method to understand the performance is key in selection of heat exchanger as highlighted by Pignotti[3].

The study by Metoui[4] highlights that the turbulence and hydraulic performance plays a key role in heat exchange. Because of the complicated nature and multiple factors involved, simulations by Computational Fluid Dynamics is suggested by Ramos[5], which allows visualizing as well as understanding the dynamic nature of the flow. The selection of the heat exchanger for compact design hence requires a turbulent based design with an effective surface area for heat exchange. The study by Mortean[6] highlights that use of a concentric based design with cross flow can improve the heat exchange effectiveness. Song[7] shows the effectiveness of a cross flow system, which is effective for a compact design during high heat transfer.

However, the study indicates that the study is limited for a cross flow type shell and tube compact heat exchanger. Hence the study aims to understand the performance of a cross flow heat exchanger using analytical, experimental and CFD technique.

2. Components Specification of Heat Exchanger

2.1. Bank of Tubes
The bank of tubes consists of three tubes in a periodic way. The flow through tube banks or packed beds in a heat exchanger has similar characteristics of both internal and external flows. The tubes are placed in a triangular manner and each tube has a diameter of 15 mm with a pitch of 10mm.

2.2. Cold Water Flow Profile
The two streams flow at right angles to each other. The hot stream flows inside the tubes arranged in a bank or bundle, and the cold stream flows through the bank in a direction at right angles to the tubes. Either one or both the streams may be unmixed. This configuration is intermediate in the effectiveness between parallel flow and counter flow exchangers. But it is often simpler to construct owing to the relative simplicity of the inlet and outlet flow ducts. The duct diameter is of 30mm

2.3. Shell
The shell used in heat exchanger has a diameter of 60mm and a length of 200mm. The ends of the shell are sealed using two end plates. The shell has two inlets for the cold water and an exit at the bottom as seen in figure 1.

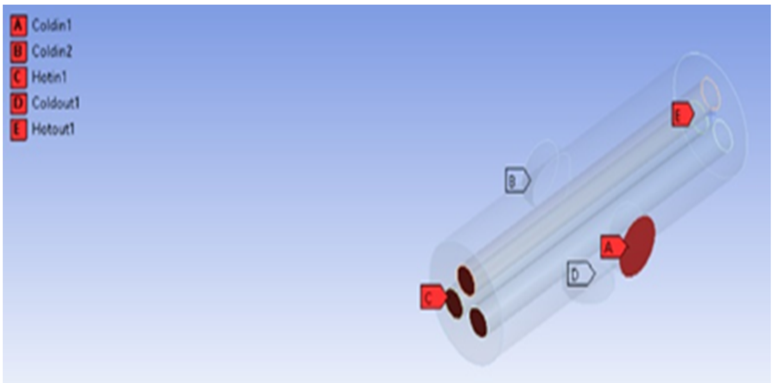

Figure 1. Schematic of the heat exchanger

3. Computational Method

3.1. Meshing of the Geometry
The given geometry is drawn to the scale by using the pre-processor of the ANSYS-FLUENT Software. The Meshing is done and grid quality of the mesh is checked for the aspect ratio and skewness. The summary of the mesh created is given in the Table I. The aspect ratio and skewness are within the

allowable limit, the mesh quality can be termed as satisfactory. The meshed quality is expected to be moderate to achieve satisfactory result as observed in Mroue[8].

Table 1. Mesh Details

Element Type	Number of Elements	Maximum Skewness	Aspect Ratio
Tetrahedral/ hybrid T- grid	1153711	0.82 in 10 Elements	4

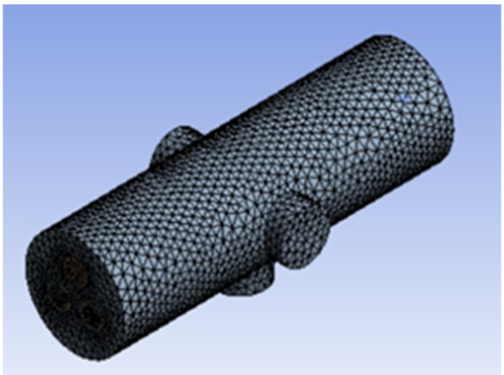

Figure 2. Meshed model of the heat exchanger

3.2. Solver Type
The solver type used in the present problem is 3-D segregated.

3.3. Boundary Condition
The boundary condition for the problem are:

3.3.1. Hot Fluid (Distilled Water):
- Mass flow rate = 0.01 m^3s^{-1} (Reynold's Number (Re) > 10^6)
- Inlet Temperature = 350K
- Turbulence Intensity = 3.5%

3.3.2. Cold Fluid (Water):
- Mass flow rate = 0.1 m^3s^{-1} (Reynold's Number (Re) > 10^7)
- Inlet Temperature = 300 K
- Turbulence Intensity = 3.5%

3.3.3. Discretization Procedure:
- Pressure connection scheme: Standard
- Momentum: Second order
- Pressure velocity coupling: SIMPLE
- Energy: Second order
- Turbulence Kinetic Energy: Second Order

4. Performance Evaluation of Heat Exchanger
To calculate, the effectiveness 'ξ' of heat exchanger, the LMTD and NTU are used together. The initial temperatures for hot and cold fluids are given.

After specifying the proper boundary conditions and initialization of the problem, the solution is iterated with segregated solver. These temperature values of outlet along with the inlet temperature are used in calculating the effectiveness of the present heat exchanger problem using Number of Transfer Units (NTU) method based on the Log Mean Temperature Difference (LMTD) method to find temperature difference.

A test rig is constructed for to perform the experimental analysis. The data provided in the simulation is used as the inlet and the outlet temperature. The temperature differences used for finding the effectiveness is obtained from the LMTD method.

5. Results and Discussion

The results are achieved after multiple iterations until a suitable convergence is achieved. After 900 – iterations the final converged solution is obtained. The solution gives the final temperatures for hot and cold fluid at the outlet are tabulated in Table 2. The results of the experiment are tabulated in Table 3. Based on the results, the effectiveness is calculated using the NTU method.

Table 2. Simulation Results

Fluid Type	Inlet Temperature (K)	Outlet Temperature (K)	Specific Heat (Cp) ($kJkg^{-1}K^{-1}$)
Fluid 1: Hot	350	331.69	4190
Fluid 2: Cold	300	328.86	4178

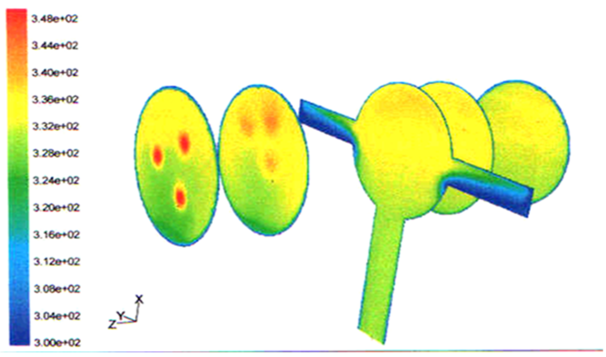

Figure 3. Temperature contour of Simulation Model

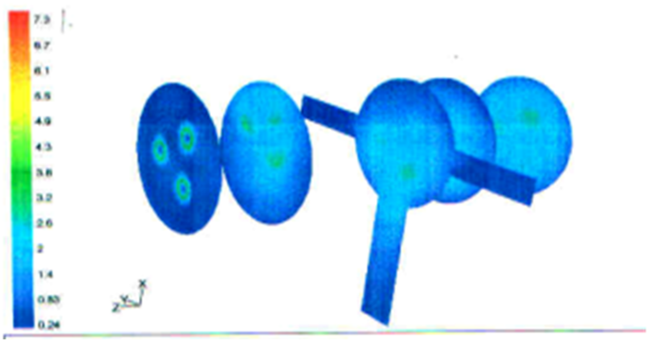

Figure 4. Turbulence Contour of Simulation Model

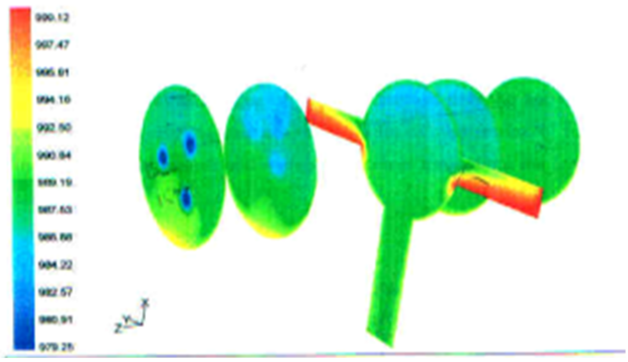

Figure 5. Density Variation in Simulation Model

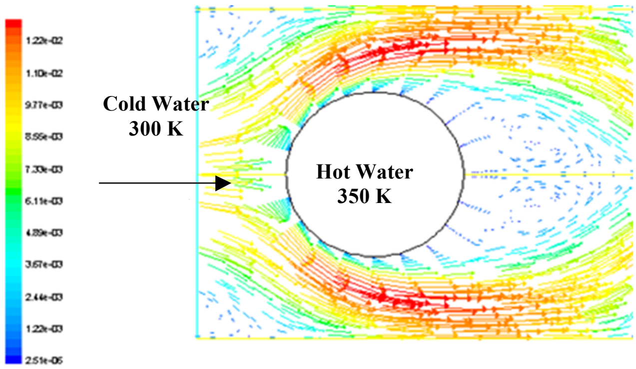

Figure 6. Temperature Contour Across the pipe

Table 3. Experimental Results

Fluid Type	Inlet Temperature (K)	Outlet Temperature (K)	Specific Heat (Cp) ($kJkg^{-1}K^{-1}$)
Fluid 1: Hot	350	333	4190
Fluid 2: Cold	300	330	4178

The result highlights that the simulation effectiveness of the Heat Exchanger is 91% against the effectiveness of the experimental method of 86%. The result supports that the cross-flow heat exchangers can be an effective method when the space is limited. The study also supports that the NTU method can be accurate in calculation of cross flow heat exchanger effectiveness.

5.1. Comments on Temperature
The distribution of temperature shows the effectiveness of cold liquid to cool the hot liquid flowing through 3-tubes. The mass flow ratio between the hot/cold liquid has been kept nearly 10 times. The equilibrium temperature distribution shown in the Figure 3 helps to understand the heat and mass transfer in a typical heat exchanger.

5.2. Comments on Turbulent Intensity
The distribution of turbulence intensity at different section of heat exchanger shows that a maximum value of 7.3% is observed near the hot tube portion. The turbulence intensity plot in Figure 4 highlights

that a lot of momentum transfer between the hot and cold liquids and within themselves throughout the length of the heat exchanger.

5.3. Comments on Fluid Density

The distribution of fluid density as observed in Figure 5 at different section of heat exchanger provides a clear indication that process of heat transfer and convection are due to density difference of water which ranged from 999.18 Kg/m3(for cold water) to 979.25 Kg/m3 (for hot water).

6. Conclusion

The value of effectiveness of present heat exchanger 'ξ' computed is equal to 91% and from the graph is 86%. The obtained value for 'ξ' from the graph is nearly tally with one obtained from the calculations. Hence, we can conclude that the solution is converged in the simulation of heat transfer. The conclusions can be drawn that the turbulence and placement of the tubes are critical in effective heat transfer. The high effectiveness also concludes that the heat exchanger can be employed in compact environments.

The study also recommends to include more physics like heat generation from shell and tube, hot liquid, radiation models, including a greater number of tubes, with different tube layouts and with different angles. Further it is suggested to use CFD tool to optimize the diameter ratio, flow ratio for a given amount of Heat transfer

Acknowledgment

The authors would like to thank Director, FCRI, Palakkad for their support and encouragement.

References

[1] Fard MH, Talaie MR and Nasr S 1 Mar 2011 Numerical And Experimental Investigation Of Heat Transfer Of Zno/Water Nanofluid In The Concentric Tube And Plate Heat Exchangers *Thermal science* **pp. 15-16.**

[2] Akhtari M, Haghshenasfard M and Talaie MR. 15 Jun 2013 Numerical and experimental investigation of heat transfer of α-Al2O3/water nanofluid in double pipe and shell and tube heat exchangers *Numerical Heat Transfer, Part A: Applications.* **63(12), pp. 941-58.**

[3] Pignotti A and Shah RK 1 May 1992 Effectiveness-number of transfer units relationships for heat exchanger complex flow arrangements *International journal of heat and mass transfer* **35(5), pp. 1275-91**

[4] El Maakoul A, El Metoui M, Abdellah AB, Saadeddine S and Meziane M 25 Dec 2017 Numerical investigation of thermohydraulic performance of air to water double-pipe heat exchanger with helical fins *Applied Thermal Engineering* **127, pp. 127-39.**

[5] Ramos J, Chong A and Jouhara H 1 Nov 2016 Experimental and numerical investigation of a cross flow air-to-water heat pipe-based heat exchanger used in waste heat recovery *International Journal of Heat and Mass Transfer* **102, pp. 1267-81.**

[6] Mortean MV, Cisterna LH, Paiva KV and Mantelli MB 5 Mar 2019 Thermal and hydrodynamic analysis of a cross-flow compact heat exchanger *Applied Thermal Engineering* **150 pp. 750-61.**

[7] Song P, Wang B, Li X and Shi W 5 May 2018 Experimental research on heat and mass transfer characteristics of cross-flow closed-type heat-source tower *Applied Thermal Engineering* **pp 135:289-303.**

[8] Mroue H, Ramos JB, Wrobel LC and Jouhara H 5 Mar 2015 Experimental and numerical investigation of an air-to-water heat pipe-based heat exchanger *Applied Thermal Engineering* **78, pp. 339-50.**