Journal of Civil Engineering and Environmental Technology
Print ISSN: 2349-8404; Online ISSN: 2349-879X; Volume 1, Number 3; August, 2014 pp. 66-70

http://www.krishisanskriti.org/jceet.html

Computational Fluid Dynamics Analysis of Shell and Tube Heat Exchanger

Gurbir Singh[1], Hemant Kumar[2]

[1]*M.Tech, Deptt. of Mechanical Engineering, Punjabi university, Patiala*
[2]*Deptt. of Mechanical Engineering, Punjabi university, Patiala*

Abstract: **In this paper, 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. The turbulence model k-ε is used for accurate results from CFD. The temperature variations are calculated from experiment for parallel and counter flow by varying the mass flow rate of fluid of 2L/min and 3L/min which is controlled by rota meter and the temperature variations are noted by the sensors attached at the inlets and outlets of tube. The solid geometry is made in SOLID WORKS software and then imported into GAMBIT which is the pre-processor of the ANSYS 13.0 for meshing the model geometry. Using the post processor FLUENT, the simulated results are computed i.e. temperature contours, pressure contours and velocity vectors. Then, simulated results are validated with the experimental values. The analysis shows that there is a difference between temperatures values computed from the experiment and the simulation by ANSYS 13.0. CFD helps to design the heat exchanger by varying the different variables very easily otherwise it is very difficult if done practically. CFD models or packages provides the contours and data which predict the performance of the heat exchanger design and are effectively used because it has ability to obtain optimal solutions and has work in difficult and hazardous conditions.**

Keywords: **Shell and tube heat exchanger, CFD, Simulation, Fluent.**

1. INTRODUCTION

Heat exchangers are one of the mostly used equipments in the process industries. Heat exchangers are used to transfer heat between two process streams. Heat exchanger is required in process which involves cooling, heating, condensation, boiling or evaporation.

1.1 Shell and tube heat exchangers: Shell and tube heat exchangers consist of a series of tubes. One set of these tubes contains the fluid that must be either heated or cooled. The second fluid runs over the tubes that are being heated or cooled so that it can either provide the heat or absorb the heat required. A set of tubes is called the tube bundle and can be made up of several types of tubes: plain, longitudinally finned, etc. Shell and tube heat exchangers are typically used for high-pressure applications. This is because the shell and tube heat exchangers are robust due to their shape. Heat exchangers are widely used in industry both for cooling and heating large scale industrial processes. The type and size of heat exchanger used can be made to suit a process depending on the type of fluid, its phase, temperature, density, viscosity, pressures, chemical composition and various other thermodynamic properties.

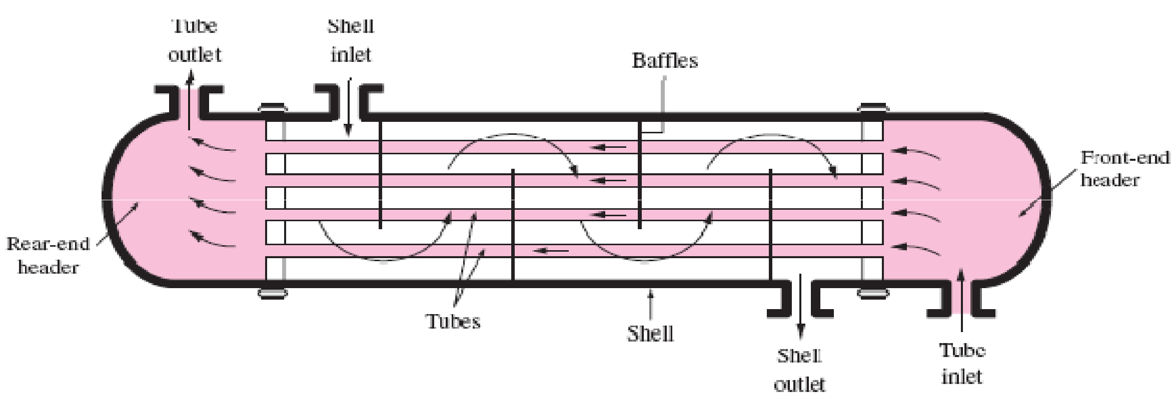


Fig: 1. Shell and tube heat exchanger

1.1.1 Parallel flow: In a parallel-flow exchanger, as the name suggests, the two fluid streams (hot and cold) travel in the same direction. The two streams enter at one end and leave at the other end. The flow arrangement of the fluid streams in case of parallel flow heat exchangers are shown in Fig. 2.1. The heat exchanger is performing at its best when the outlet temperatures are equal.

1.1.2 Counter flow: In counter flow heat exchangers the fluids enter the exchanger from opposite ends. Counter flow heat exchangers are more efficient than parallel flow heat exchangers because they create a more uniform temperature difference between the fluids, over the entire length of the fluid path. Counter flow heat exchangers can allow the cold fluid to exit with a higher temperature than the exiting hot fluid. The flow arrangement for such a heat exchanger is shown schematically in Fig. 2.2.

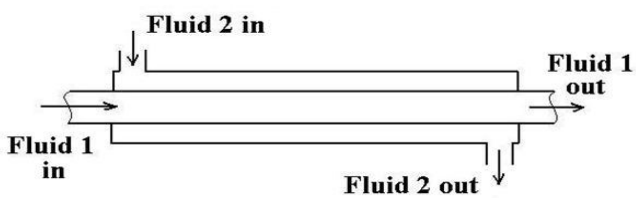


Fig: 2.1 Parallel flow HE

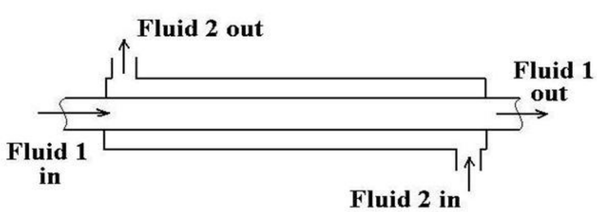


Fig: 2.2 Counter flow HE

2. Experimental details: This setup is available in heat transfer lab of Punjabi university, Patiala shown in fig 3. The required parameters will be calculated in heat transfer lab through following heat exchanger. After that simulation is done in ANSYS. CFD analysis of heat exchanger will be done in ANSYS V.13 which is a useful CFD code currently used by most of the researchers. As explained earlier, grid generation will be done in GAMBIT which is pre processor of ANSYS. This includes the setting boundary conditions, defining fluid properties and refining the grid. Then mesh file will be exported to FLUENT which works as solver for ANSYS. The CFD analysis will be done in FLUENT using difference models. Then temperature contours, velocity vectors and pressure drop contours will be obtained in post processor which is FLUENT itself. FLUENT provides complete mesh flexibility, solving the flow problems. These results are validated with the experimental results.

Fig: 3. Apparatus

Dimensions:-
Length of pipe = 1610 mm
Inner tube: Material =SS of inner diameter = 9.5 mm and outer diameter = 12.7 mm
Outer tube**:** Material GI of inner diameter = 28 mm and outer diameter = 33.8 mm
Minimum distance of cold water nozzle from pipe end = 20 mm
Cold water nozzle inner diameter = 16.8 mm

The experiment is performed for parallel flow and counter flow. In parallel flow, the flow of hot and cold water is in one direction whereas in counter flow, the flow is in opposite direction. The flow rate of hot and cold water is controlled by rota meter. The readings are taken for the flow rate of 2L/min and 3L/min for both parallel and counter flow as shown in table 1 and table 2 respectively.

Table1. Calculation for parallel flow

Temp. sensor no	**Temp.**	**Flow rate**	
		2L/min	3L/min
S_1	T_{hi}	57.2°C/330.2K	57.6°C/330.6K
S_2	T_{ho}	47.5°C/320.5K	48.8°C/321.8K
S_3	T_{ci}	20.8°C/293.8K	20.8°C/293.8K
S_4	T_{co}	28.8°C/301.8K	28.2°C/301.2K

Table2. Calculation for counter flow

Temp. sensor no.	Temp.	Flow rate	
		2L/min	3L/min
S_1	T_{hi}	57.4°C/330.4K	57.2°C/330.2K
S_2	T_{ho}	47.1°C/320.1 K	48.8°C/321.8K
S_3	T_{ci}	21.1°C/294.1K	21.0°C/294.0K
S_5	T_{co}	29.5°C/302.5K	28.4°C/301.4K

Where,
T_{hi} = Inlet temperature of hot water, T_{ho}= outlet temperature of hot water, T_{ci} = Inlet temperature of cold water,
T_{co} = outlet temperature of cold water.
S_1, S_2, S_3, S_4, S_5 = Temperature sensors at inlets and outlets of water.

3. Simulation: The simulation of heat exchanger is done in ANSYS which is useful code used by researchers these days. ANSYS includes the pre processor called GAMBIT in which grid or mesh generation is done. Meshing means dividing or integrating the large volume into small finite elements. In computational fluid dynamics, meshing is a discrete representation of the geometry that is involved in the problem. Fluid dynamics simulations require very high-quality meshes in both element shape and smoothness of sizes changes.

3.1 Meshing: Initially the part solid of heat exchanger is made in solid works. In this, only solid geometry is formed as shown in fig. 4.

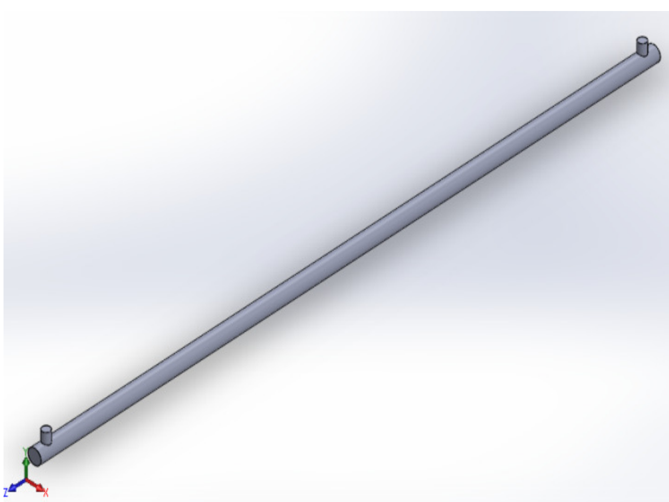


Fig: 4. Solid geometry of heat exchanger

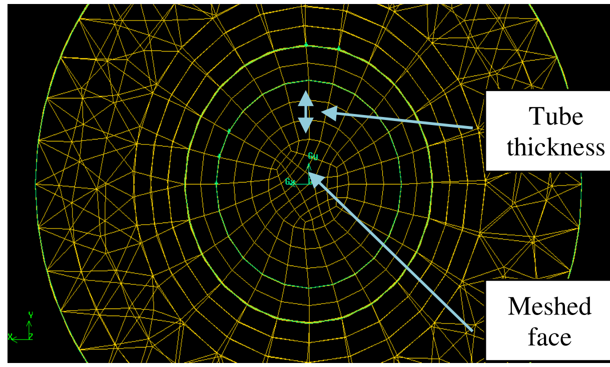


Fig: 5. Meshed face of tube

After that the geometry is imported to GAMBIT where grid generation is done. In GAMBIT, the parasolid imported is a single volume. But the heat exchanger used is shell and tube type i.e. there is a tube inside the shell in which hot water is flowing and cold water is flowing inside the shell. So the face of hot water tube of inner diameter 9.5mm and outer diameter 12.7mm is made and another volume is made by sweeping the faces and splitting it from the first volume. Another volume is formed on the both sides of the tube of same diameter i.e. at the inlet and the outlet end. The edge of tube has meshed by giving the interval count of 24. The boundary layer function is used to get the precise elements near the wall of the tube because the temperature difference and heat transfer has varied more there. Then face of tube has meshed with quadrilateral elements of pave type as shown in fig.5.

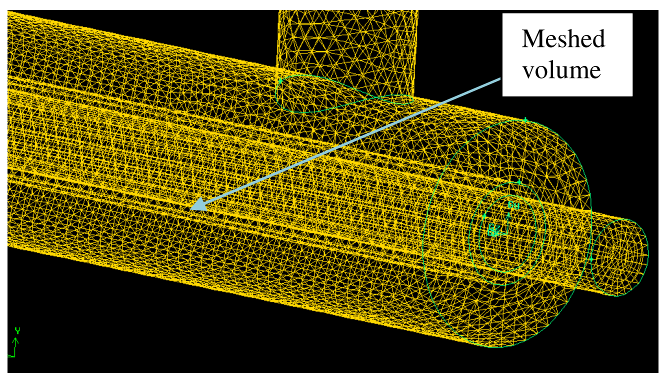


Fig: 6. Meshed volume of tube

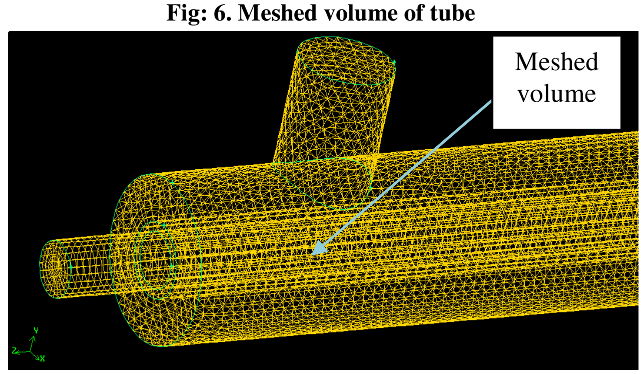
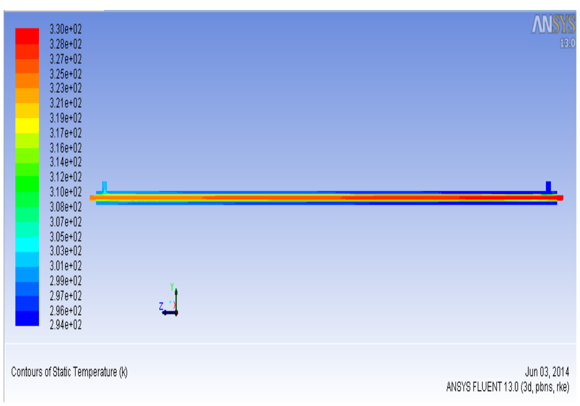


Fig: 7. Meshed volume of shell

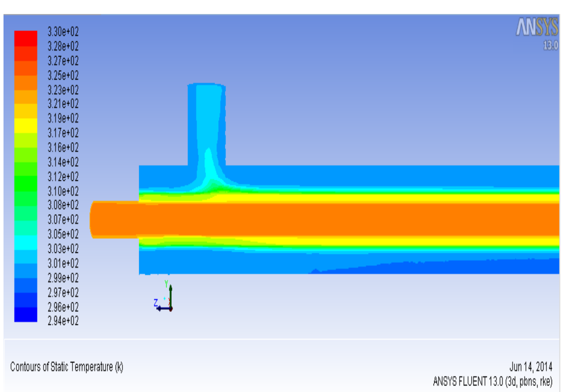
After that, volume of tube and thickness of the tube along the length, meshing has done with hexahedral elements of copper type as shown in fig.6.

After meshing the tube, shell has to be meshed. Again the boundary layer is created on the outside edge of the tube to mesh the face near the wall of the tube. Then the volume of the shell is meshed by tetrahedral elements of type. In this step the shell which includes the volume outside the surface of the tube and inlet, outlet of the cold water has fully meshed as shown in fig.7. In mesh generation 199725 nodes and 486394 elements are formed. Also 12 vertices, 12 edges, 15 faces and 3 volumes are created.

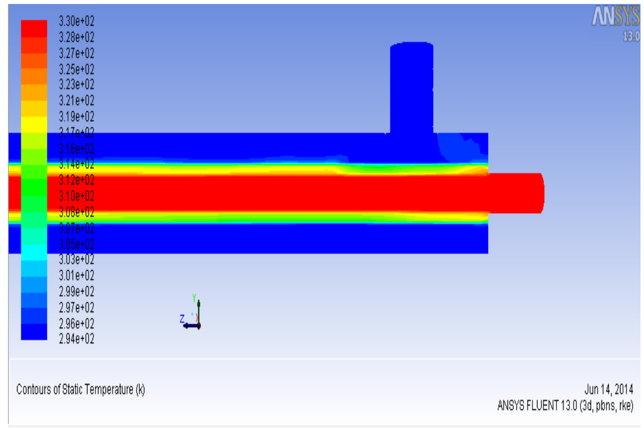
Now the mesh generated is imported to FLUENT which is a post processor of ANSYS. In FLUENT, the temperature contours, pressure contours and velocity vectors are obtained for parallel and cross flow. After importing the mesh, it is scaled in 'm' units and then materials are defined: fluid as water-liquid and solid as steel. The temperature contours for parallel flow obtained are shown in fig 8(a) concluded that the temperature variation occur at the inlet and outlet of the tubes in fig 8(b) and fig 8(c) respectively. Similarly the contours for counter flow are taken by varying the mass flow rate.



(a)



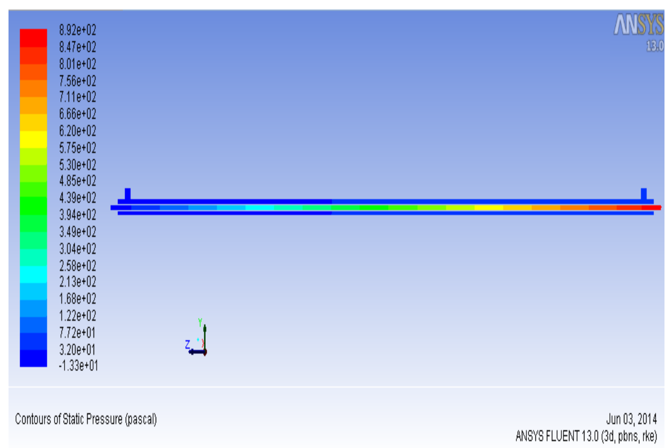
(b)



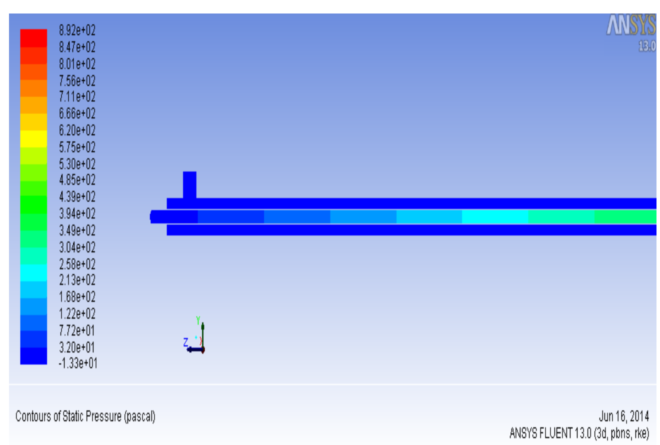
(c)

Fig: 8.Temperature contours

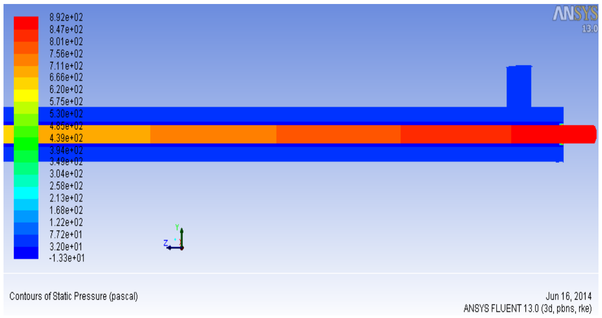
Similarly the pressure contours and velocity vectors are obtained shown in fig 9 and fig 10 respectively.



(a)

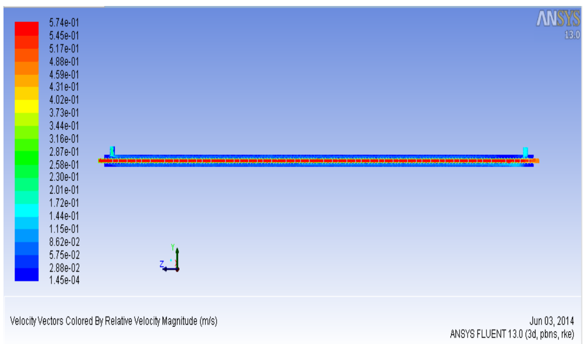


(b)

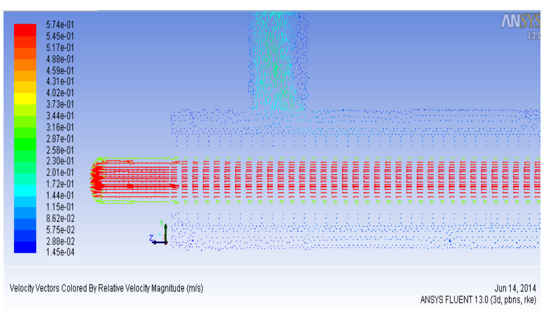


(c)

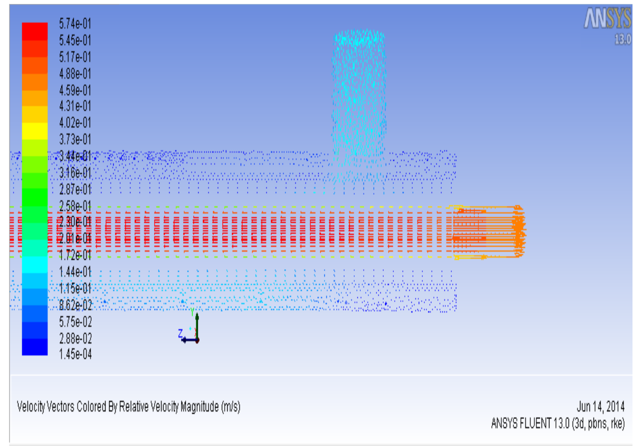
Fig: 9. Pressure contours



(a)



(b)



(c)

Fig: 10. Velocity vectors

It is concluded that the difference between the temperature of the experimental values and the simulated values are fair agreement i.e. it is accepted. CFD helps to design the heat exchanger by varying the different variables very easily otherwise it is very difficult if done practically.CFD models or packages provide the contours and data which predict the performance of the heat exchanger design.

REFERENCES

[1] Versteeg, H.K., Malalasekera, W., An introduction to computational fluid dynamics: the finite volume method.

[2] USMAN UR REHMAN, “Heat Transfer Optimization of Shell and- Tube Heat Exchanger through CFD Studies” 2011.

[3] K. Sudhakara Rao, “analysis of flow maldistribution in tubular heat exchangers by fluent”, 2007.

[4] Swapnaneel Sarma, D.H.Das, “CFD Analysis of Shell and Tube Heat Exchanger using triangular fins for waste heat recovery processes” International Journal (ESTIJ), ISSN: 2250-3498, Vol.2, No.6, December 2012.

[5] KHAIRUN HASMADI OTHMAN, “CFD simulation of heat transfer in shell and tube heat exchanger”, APRIL 2009.