A Review on CFD Analysis of Shell And Tube Heat Exchanger

Dr.S Jayakrishna[1], Ayub Ashwak[2], B.Harish Raj[3]
[1, 2, 3] Dept of Mech. Engineering
[1, 2, 3] Holy Mary Institute of Technology & Science, Hyderabad, INDIA

Abstract- *In this paper, the shell and tube warmness exchanger is considered wherein warm water is flowing inner one tube and bloodless water runs over that tube. Computational fluid dynamics method that's a pc based totally analysis is used to simulate the warmth exchanger concerning fluid float, warmth transfer. CFD clear up the whole warmth exchanger in discrete factors to discover the temperature gradients, stress distribution and velocity vectors. The turbulence model okay-ε is used for correct outcomes from CFD.*

The temperature versions are calculated from experiment for parallel and counter drift via varying the mass waft charge of fluid of 2L/min and 3L/min that's controlled by using rota meter and the temperature variations are noted with the aid of the sensors attached at the inlets and shops of tube. The stable geometry is made in SOLID WORKS software program and then imported into GAMBIT that's the pre-processor of the ANSYS thirteen.Zero for meshing the version geometry. Using the publish processor FLUENT, the simulated results are computed i.E. Temperature contours, strain contours and pace vectors. Then, simulated outcomes are tested with the experimental values. The evaluation suggests that there may be a distinction between temperatures values computed from the experiment and the simulation by using ANSYS thirteen.Zero. CFD helps to design the warmth exchanger by means of various the unique variables very easily in any other case it is very hard if accomplished practically. CFD fashions or applications affords the contours and facts which are expecting the performance of the warmth exchanger layout and are effectively used as it has potential to achieve most fulfilling solutions and has paintings in hard and dangerous conditions.

Keywords- Shell and tube heat exchanger, CFD, Simulation, Fluent.

I. INTRODUCTION

Heat exchangers are one of the primarily used equipments inside the procedure industries. Heat exchangers are used to transfer warmness among manner streams. Heat exchanger is needed in system which includes cooling, heating, condensation, boiling or evaporation.

1.1 Shell and tube warmness exchangers: Shell and tube heat exchangers encompass a series of tubes. One set of those tubes carries the fluid that should be both heated or cooled. The 2d fluid runs over the tubes which are being heated or cooled so that it may either offer the warmth or absorb the heat required. A set of tubes is known as the tube bundle and can be made of numerous varieties of tubes: simple, longitudinally finned, and so forth. Shell and tube warmness exchangers are typically used for high-strain packages. This is due to the fact the shell and tube warmness exchangers are strong because of their shape. Heat exchangers are broadly used in industry each for cooling and heating big scale industrial procedures. The type and length of warmth exchanger used may be made to suit a process depending on the type of fluid, its phase, temperature, density, viscosity, pressures, chemical composition and numerous other thermodynamic houses.

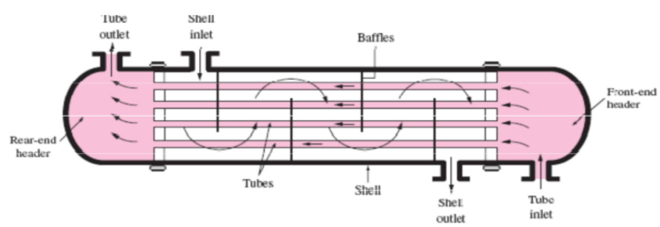


Fig: 1. Shell and tube heat exchanger

1.1.1 Parallel go with the flow: In a parallel-drift exchanger, as the call indicates, the two fluid streams (hot and co ld) tour in the same course. The streams enter at one stop and leave at the opposite quit. The drift association of the fluid streams in case of parallel waft heat exchangers are sh own in Fig. 2.1. The heat exchanger is performing at its exceptional when the opening temperatures are same.

1.1.2 Counter go with the flow: In counter go with the flow warmness exchangers the fluids input the exchanger from opposite ends. Co unter drift heat exchangers are greater efficient than para llel glide warmness exchangers due to the fact they create a extra unif orm temperature difference between the fluids, over the enti re period of the fluid path. Counter flow heat exchangers ca n permit the cold fluid to exit with a higher temperature than the exiting hot fluid. The flow association for the sort of warmth exchanger is proven schematically in Fig. 2.2

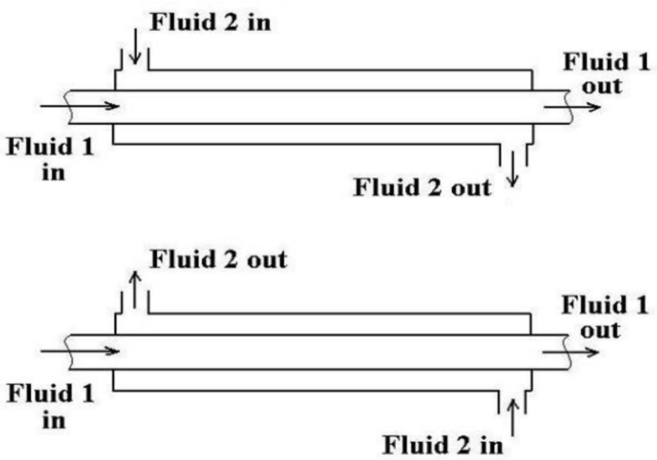


Fig: 2.2 Counter flow HE

II. EXPERIMENTAL INFO

This setup is availabl e in warmness switch lab of Punjabi university, Patiala shown in fig three. The required parameters could be calculated in warmth trans fer lab through following warmness exchanger. After that simula tion is accomplished in ANSYS. CFD evaluation of heat exchanger could be executed in ANSYS V.Thirteen that is a beneficial CFD code currently utilized by most of the researchers. As explained in advance, grid technology will be done in GAMBIT that is pre processor of ANSYS. This consists of the setting boundary situations, defining fluid homes and refining the grid. Then mesh report will be exported to FLUENT which works as solver f or ANSYS. The CFD evaluation might be completed in FLUENT the usage of distinction models. Then temperature contours, velocity vectors and pressure drop contours might be received in submit processor that is FLUENT itself. FLUENT offers entire mesh flexibility, solving the flow pro blems. These results are validated with the experimental results

Fig: 3. Appa ratus

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 flo w, 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.

The experiment is completed for parallel float and counter waft. In parallel glide, the drift of hot and bloodless water is in one path whereas in counter flo w, the go with the flow is in contrary path. The waft charge of hot and bloodless water is controlled through rota meter. The readings are taken for the flow price of 2L/min and 3L/min for each parallel and counter go with the flow as shown in table 1 and table 2 respectively.

Table1. Calculation fo r parallel flow

Temp. sensor no	Temp.	Flow rate	
		2L/mi n	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 rat e	
		2L/min	3L/min
S_1	T_{hi}	57.4°C/330.4K	5 7.2°C/330.2K
S_2	T_{ho}	47.1°C/320.1 K	4 8.8°C/321.8K
S_3	T_{ci}	21.1°C/294.1K	2 1.0°C/294.0K
S_5	T_{co}	29.5°C/302.5K	2 8.4°C/301.4K

Where,
T_{hi} = Inlet temperature of hot water, T_{ho}= outl et 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.

III. SIMULATION

The simulation of heat excha nger is achieved in ANSYS which is beneficial code used by researc hers in recent times. ANSYS includes the pre processor known as GA MBIT in which grid or mesh era is finished. Meshing manner dividing or integrating the big quantity into small finite elements. In computational fluid dynamics, meshing is a discrete illustration of the geometry this is involved in the problem. Fluid dynamics simulations require very excessive - great meshes in both detail shape and smoothness of sizes modifications.

3.1 Meshing: Initially the element stable of heat exchanger is made in solid works. In this, simplest solid geometry is f ormed as shown in fig. Four

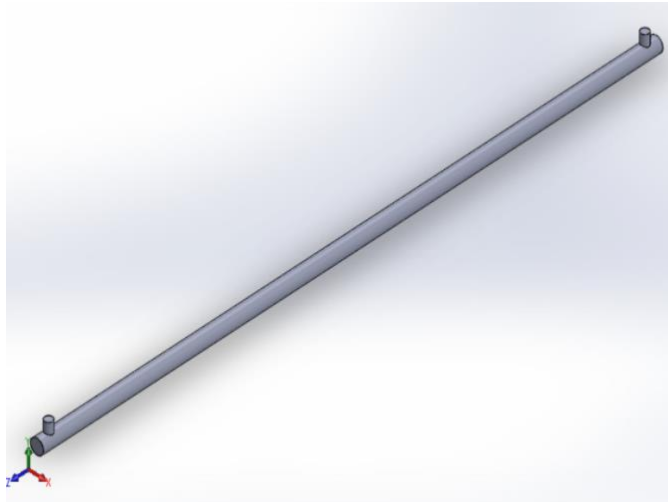


Fig: 4. Solid geometry of heat exchanger

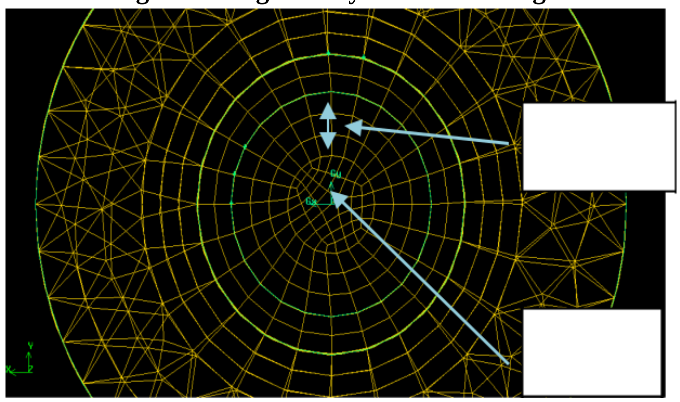


Fig: 5. Meshed face of tube

After that the geometry is import ed to GAMBIT in which grid technology is performed. In GAMBIT, the parasolid imported is a unmarried quantity. But the heat exchanger used is shell and tube type i.E. There is a tube within the shell in which hot water is flowing and bloodless water is flowing in the shell. So the face of warm water tube of inner diameter nine.5mm and outer diameter 12.7mm is made and any other volu me is made via sweeping the faces and splitting it from the first extent. Another volume is shaped on the both sides of the tu be of equal diameter i.E. At the inlet and the hole stop. The edge of tube has meshed by way of giving the c language count number of 24. Th e boundary layer function is used to get the best elements close to the wall of the tube due to the fact the temperature differen ce and heat transfer has numerous extra there. Then face of tube has meshed with quadrilateral elements of pave type as proven in fig.Five

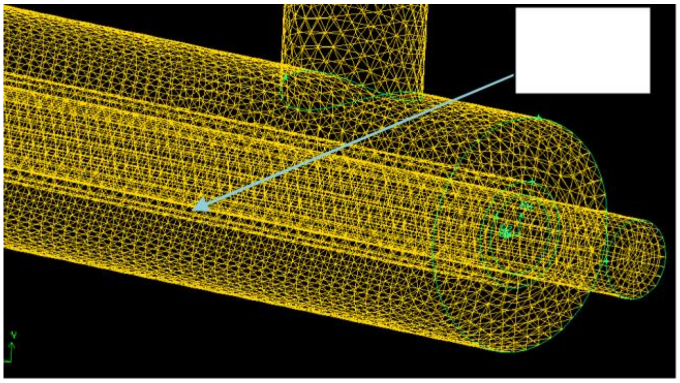
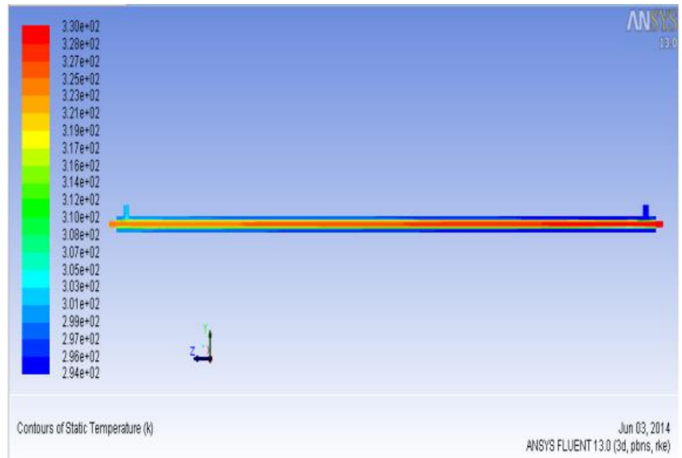


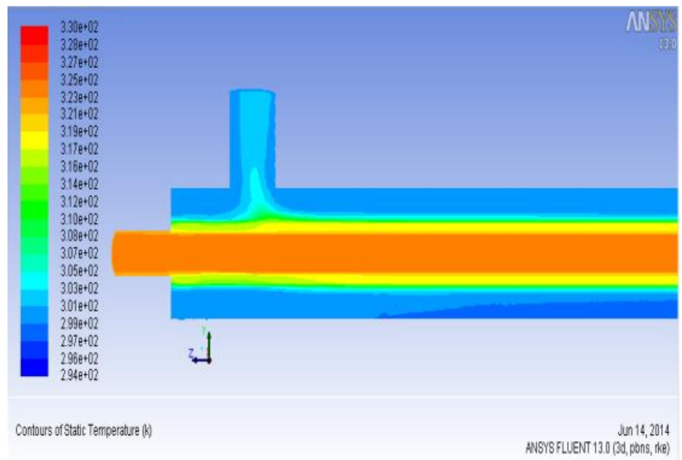
Fig: 6. Meshed vol ume of tub After

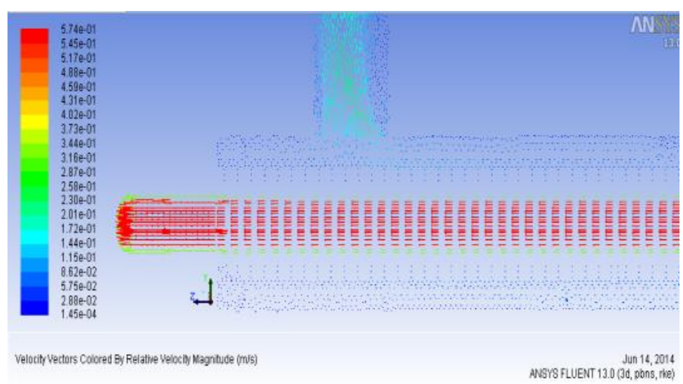
After that, extent of tube and thickness of th e tube along the duration, meshing has completed with hexahedral factors of copper kind as proven in fig.6.

After meshing the tube, shell must be meshed. Again the boundary layer is created at the out of doors edge of the tube to mesh the face near the wall of the tube. Then the extent of the shell is meshed by way of tetrahedral factors of type. In this step the shell which incorporates the volume out of doors t he floor of the tube and inlet, outlet of the cold water has fully meshed as shown in fig.7. In mesh era 199725 n odes and 486394 factors are shaped. Also 12 vertices, 12 edg es, 15 faces and three volumes are created.

Now the mesh generated is imported to FLUENT that's a post processor of ANSYS. In FLUENT, the temperature contours, pressure contours and pace vectors are received for parallel and move glide. After importing the mesh, it's miles scaled in 'm' gadgets after which materials are d efined: fluid as water-liquid and strong as metallic. The temperature contours for parallel waft obtained are shown in fig eight(a) co ncluded that the temperature variant arise at the inlet and o utlet of the tubes in fig eight(b) and fig eight(c) respectively. Similarly the contours for counter glide are taken with the aid of various the mass float fee. that, extent of tube and thickness of th e tube along the duration, meshing has completed with hexahedral factors of copper kind as proven in fig.6After meshing the tube, shell must be meshed. Again the boundary layer is created at the out of doors edge of the tube to mesh the face near the wall of the tube. Then the extent of the shell is meshed by way of tetrahedral factors of type. In this step the shell which incorporates the volume out of doors t he floor of the tube and inlet, outlet of the cold water has fully meshed as shown in fig.7. In mesh era 199725 n odes and 486394 factors are shaped. Also 12 vertices, 12 edg es, 15 faces and three volumes are created.







It is concluded that the difference between the temperature of the experimental values and the simulated values are honest agreement i.E. It's miles widely wide-spread. CF D helps to layout the warmth exchanger with the aid of various the diffe lease variables very effortlessly in any other case it is very hard if executed practically.CFD fashions or packages offer the contours a nd information which expect the overall performance of the warmth exchanger layout.

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, "C FD 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 OTHMA N, "CFD simulation of heat transfer in shell and tube heat exc hanger", APRIL 2009.