International Journal of Engineering Research ISSN: 2319-6890)(online),2347-5013(print)
Volume No.5 Issue: Special 6, pp: 1129 -1254 20 May 2016

CFD analysis of shell and tube heat exchanger

[1]Praveen Kumar Kanti, [2]Karthika.U.P, Sabeer Ali,Sanath Kumar.N ,Shyam Chandran C

UG scholar, Department of mechanical, Jyothi Institute Of Technology, Tataguni, Bangalore-560062, India

Asst.Prof, Department of Mechanical, Jyothi Institute Of Technology, Tataguni, Bangalore-560062, India

Email: karthika71295@gmail.com, praveenkanti87@gmail.com, sanathkumarn@outlook.com

Abstract--- **Heat exchangers are used to transfer heat from fluid at high temperature to fluid at lower temperature. Heat exchangers are used in industrial purposes in chemical industries, nuclear power plants, refineries, food processing, etc. Sizing of heat exchangers plays very significant role for cost optimization. Also, efficiency and effectiveness of heat exchangers is an important parameter while selection of industrial heat exchangers. Methods for improvement on heat transfer have been worked upon for many years in order to obtain high efficiency with optimum cost. In this paper, we are analysing shell and tube heat exchanger without baffle plates by changing their outer material .The calculations and simulations are done for counter flow of the heat exchanger.**

Keywords--- **Shell and Tube Heat Exchanger, CFD-computational fluid dynamics, Heat Transfer Characteristics, effectiveness.**

I. INTRODUCTION

A heat exchanger transfers energy from one fluid to another across a solid surface by convection and conduction. Heat exchangers are used in power plants, nuclear reactors, refrigeration and air conditioning systems, automotive industries, heat recovery systems, chemical processing, and food industries. The design of a new heat exchanger (HE) is referred to the sizing problem, means it includes construction type, flow arrangement, tube and shell material, and physical size which has to meet the specified heat transfer and pressure drop rating of existing heat exchanger. We are going to simulate the results for present material of shell and tube heat exchanger and change the material, then compare the results of both and study which is best .

Shell and tube heat exchanger consist bundle of tubes enclosed with in cylindrical shell one fluid pass through the tubes and second fluid flows between the tube and shells. The basic components of a shell and tube heat exchangers are tubes, tube sheets, shell and shell-side nozzles, tube side channels and nozzles, channel covers, pass divider, baffles etc .Most commonly used STHE have large heat transfer surface area-to-volume ratios to provide high heat transfer efficiency in comparison with others. They are mechanically rugged enough to withstand the fabrication stresses and normal operating conditions. They can be easily cleaned and the failure parts like gaskets and tubes can be easily replaced. They offer greater flexibility of mechanical features to withstand any service requirement. They are manufactured easily for a large variety of sizes and flow configurations. They can operate at high pressures and high temperature. They can be employed for processes which require large quantities of fluid to be heated or cooled. In our case, we have heat exchanger with stainless steel as outer shell and only one tube of copper as inner tube and no baffle plates.

II. MATERIALS AND METHODOLOGY

The present work begins with the CFD analysis of shell and finned tube heat exchanger for counter flow, in order to see the effect of temperature rise and pressure drop along the length of the finned tube and the shell. The cold water has been used as the shell side fluid and hot water in the tube side fluid for heat transfer analysis. The hot fluid in the tube transfer heat to the cold fluid (water) that is flowing through the shell.

Part number	Part Of The Model	State Type
1	Inner Fluid	Fluid
2	Inner Pipe	Solid
3	Outer Fluid	Fluid
4	Outer Pipe	Solid

CFD provides the flexibility to change design parameters without the expense of hardware changes. It therefore costs less than laboratory or field experiments, allowing engineers to try more alternative designs than would be feasible otherwise. It also reduces design cycle time and cost by optimizing through computer predictions and provides higher level of confidence in prototype or field installed performance. Moreover it investigates and understands the "why" for existing problem or new equipment. In the light of the above discussion, the present work has been taken up aiming at achieving the following objectives.

- To perform a CFD analysis of shell and tube heat exchanger using FLUENT 15.0 after modelling the heat exchanger with CFD Design moduler 15.0.
- To determine the effectiveness and temperature rise in the heat exchanger, for counter flow with copper as inner material and stainless steel as outer material.
- Then, change the outer shell material with galvanized iron maintaining the inner material

same and determine the effectiveness and temperature rise.

- To compare which material gives best heat transfer characteristics and effectiveness.
- The below table represents the different cases for the theoretical calculations and CFD analysis.

	Inner Material	Outer Material
Case 1	Copper	Stainless Steel (SS)
Case 2	Copper	Galvanized Iron (GI)

III. CFD ANALYSIS

Computational fluid dynamics (CFD) study of the system starts with the construction of desired geometry and mesh for modelling the dominion. Generally, geometry is simplified for the CFD studies. Meshing is the discretization of the domain into small volumes where the equations are solved by the help of iterative methods. Modelling starts with the describing of the boundary and initial conditions for the dominion and leads to modelling of the entire system. Finally, it is followed by the analysis of the results, discussions and conclusions.

Pre –Processor

1. Geometry:

Heat exchanger is built in the ANSYS workbench design module. It is a counter-flow heat exchanger. First, the fluid flow (fluent) module from the workbench is selected. The design modeller opens as a new window as the geometry is double clicked

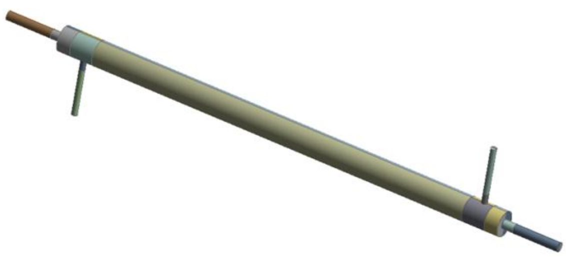


Table 1 Naming of various parts of the body with state type

2. Mesh:

Initially a relatively coarser mesh is generated. This mesh contains mixed cells (Tetra and Hexahedral cells) having both triangular and quadrilateral faces at the boundaries. Care is taken to use structured hexahedral cells as much as possible. It is meant to reduce numerical diffusion as much as possible by structuring the mesh in a well manner, particularly near the wall region.

A. Solver

1. Problem Setup

The mesh is checked and quality is obtained. The analysis type is changed to Pressure Based type. The velocity formulation is changed to absolute and time to steady state.

2. Models

Energy is set to ON position. Viscous model is selected as "k-ε model (2 equations). Radiation model is changed to Discrete Ordinates.

3. Materials

The create/edit option is clicked to add water-liquid and copper and stainless steel and galvanized iron to the list of fluid and solid respectively from the fluent database.

4. Cell zone conditions

The parts are assigned as water and copper and stainless steel or galvanized iron as per fluid/solid parts.

5. Boundary Conditions

Boundary conditions are used according to the need of the model. The inlet and outlet conditions are defined as mass flow inlet and pressure outlet. As this is a counter-flow with two tubes so there are two inlets and two outlets. The walls are separately specified with respective boundary conditions. No slip condition is considered for each wall. Except the tube walls each wall is set to zero heat flux condition. The details about all boundary conditions can be seen in the table as given below.

Inlet hot fluid temperature= 55.8 C

Inlet cold fluid temperature=31.3 C

Hot fluid flow rate=0.04 kg/s

Cold fluid flow rate=0.0525 kg/s

6. Solution Methods

The solution methods are specified as follows:

Scheme = Simple

Gradient = Least Square Cell Based

Pressure = Standard

Momentum = Second Order Upwind

Turbulent Kinetic Energy = Second Order Upwind

Turbulent Dissipation Rate = Second Order Upwind

7. Solution Control and Initialization

Under relaxation factors the parameters are

Pressure = 0.3 Pascal

Density = 1 kg/m3

Body forces = 1 kg/m2s2

Momentum = 0.7 kg-m/s

Turbulent kinetic energy = 0.8 m2/s2

Then the solution initialization method is set to Hybrid Initialization whereas the reference frame is set to Relative cell zone. The inner inlet is selected from the compute from drop down list and the solution is initialized

.

8. Measure of Convergence

It is tried to have a nice convergence throughout the simulation and hence criteria is made strict so as to get an accurate result. For this reason residuals are given as per the table that follows.

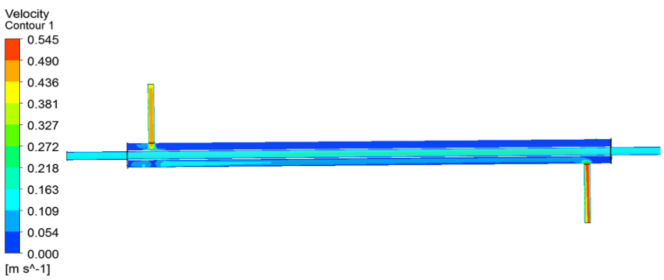
Variable	Residual
x-velocity	10-6
y-velocity	10-6
z-velocity	10-6
Continuity	10-6
Specific dissipation energy/ dissipation energy	10-5
Turbulent kinetic energy	10-5
Energy	10-7

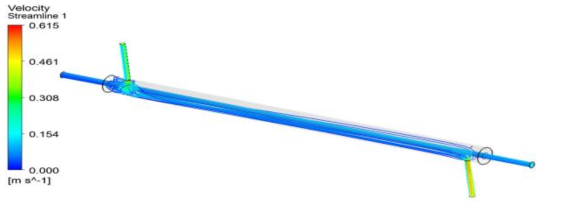
9. Run Calculation

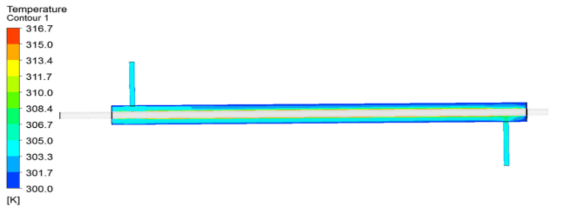
The number of iteration is set to 1000 and the solution is calculated and various contours, vectors and plots are obtained.

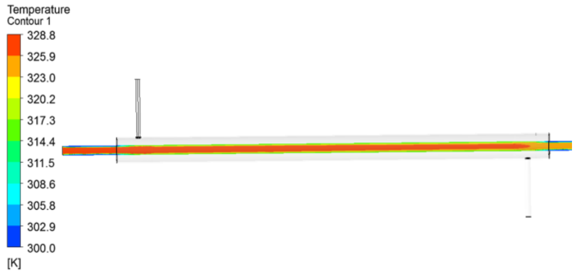
B. Post-Processor

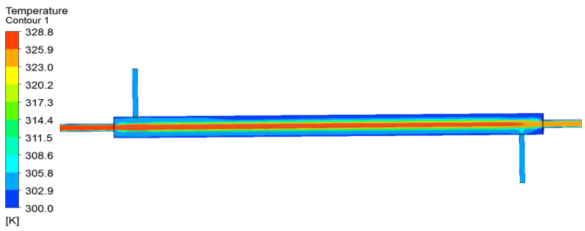
The results which are obtained is represented by various contours, vectors and plots. It is shown below











IV. Results and Tables

The results which are obtained for both the materials (stainless steel and Galvanized iron) is simulated. The experimental and simulation results are shown and compared in the table below.

	Flow rate(kg/s)	Flow rate(kg/s)	Temperature (hot fluid)			Temperature (cold fluid)			Effectiveness (exp) (%)	Effectiveness (CFD) (%)
	Hot fluid	Cold fluid	T1 Inlet	T2 Outlet (exp)	T2 Outlet (CFD)	T1 Inlet	T2 Outlet (exp)	T2 Outlet (CFD)		
Case1. (S.S)	0.04	0.0525	55.8	42.5	44.6	31.3	34.5	36.8	62	56
Case 2. (G.I)	.04	.0525	55.8	38.6	40.2	31.3	33	34	42	38

The results of both experimental and simulation are compared in the above table. The results match with a negligible percentage of error.

V. Conclusion

The comparison of Stainless Steel and Galvanized Iron reveal that Stainless Steel is a better outer material when coupled with inner material of Copper for a heat exchanger. The effectiveness and efficiency of Stainless Steel is better than Galvanized Iron by

VI. ACKNOWLEDGEMENT

We would like to acknowledge Jyothy Institute Of Technology, Tataguni, Bangalore for giving a good support. We would like to thank the principal and HOD of my institution. We would also like to thank Asst.Prof. Praveen Kumar Kanti, for guiding us doing this. Our gratitude also extends to our friends.

References

i. Rajat gupta (2013) in " Performance Analysis of a Helical Savonius Rotor with Shaft at 45° Twist Angle Using CFD".
ii. Amir Mokaramian (20130 in "Fluid Flow Investigation through Small Turbodrill for Optimal Performance".
iii. Swapnaneel Sarma (2012) in "CFD Analysis of Shell and Tube Heat Exchanger using triangular fins for waste heat recovery processes".
iv. Chiu-Fan Hsieh1 & Hao -Yu Cheng (2014) made an attempt of " Effects of Various Geometric Designs on the Flow Characteristics of a Triangular Rotary Engine"
v. Dr. Nirvesh Mehta (2013) studied the "CFD Analysis of gear box".
vi. Gita R. Rohit (2013) made an attempt to "CFD Analysis of radiator".
vii. H.Mohamed Niyaz (2013) studied the "CFD Analysis diesel engine".
viii. Amit Chauhan (2013) made an attempt "to study CFD Based Thermo-Hydrodynamic Analysis of Circular Journal Bearing".
ix. Syed Saleem Pasha (2015) work on "CFD
x. Study of the Fluid Flow Dynamic Analysis on I.C Engine using Ansys".
xi. Mr. Jekim (2014) studied "effect of CFD Analysis Of Centrifugal Pump Impeller".
xii. Jiyuan tu text book on "Computational fluid dynamics- A practical approach".
xiii. John D. Anderson book on "Computational fluid dynamics- The basics with applications".