International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 3 Issue 7, July - 2014

CFD Analysis of a Pipe in a Heat Exchanger

G. S. L. Swathi[1],
M.Tech Scholar, Department of Mechanical Engineering,
Kakinada Institute of Technology and Science, Divili,
Andhra Pradesh, India.

V. Subrahmanyam[2],
Associate Professor, Department of Mechanical Engineering, Kakinada Institute of Technology and Science, Divili, Andhra Pradesh, India.

***Abstract*: The application of heat exchanger are found in automobiles, power plants, refrigerators and air conditioners etc. The main objective of the work is to have material of the Heat Exchanger (Al & Cu), so that the heat transfer rate should be more and also deals with the pressure variations, velocity contours and temperature distributions. Two phase circular cross section geometry was developed in Gambit and Computational Fluid Dynamics calculations using K-Epsilon model were employed in Fluent software. This simulation gives the values of pressure, temperature, heat transfer rate and velocity at various sections of the pipe in which water as a fluid and air as coolant flowing out side. A comparison was made with Copper and Aluminium as the material of the pipe. The numerical results were validated against experimental data from the literature and were found to be good.**

Keywords: Gambit, Fluent, K-epsilon model.

1. INTRODUCTION

Cooling systems like radiators, condensers and evaporators have a significant effect on the efficiency of the machine. Especially a radiator exhausting heat from the engine is closely related to the engine performance and mechanical failure.

To enhance the efficiency of a Heat Exchanger, various studies have to be done on the additional devices such as a rectangular fin, a plate, a circular tube, a flat tube, an elliptic tube and a louver fin. Among the various types of Heat Exchangers circular fins used more often. The heat transfers that occur within the machine is extremely important for the proper operation. Most of the materials used for the machine components may not be able to withstand the high temperature and may damage.

So in order keep the engines or machines in proper operating conditions, cooling system has to use Heat Exchanger with water as a cooling medium for dissipating heat from the engine. In the Heat Exchangers the medium is fluid i.e., coolant is used. The main aim of Heat Exchanger is to decrease the temperature so as not to melt the machine parts. The coolant will take the heat and flows through the Heat Exchanger, in which heat transfer takes place by air or water flow over the Heat Exchanger. Finally the cooled down fluid from the outlet of the Heat Exchanger enters into the engine or machine again. This process continues throughout working of the machine.

In this paper the analysis was done on very small area of the Heat Exchanger and the effect of various operating parameters such as coolant inlet, pressure, mass flow rate and heat transfer rate are studied as well.

2. PROCEDURE

The current study used FLUENT, to solve the balance equation using control volume approach. These equations are solved by converting the complex partial differential equations into simple algebraic equations. The simple geometry is done in the GAMBIT software, a fine meshing is done by using successive ratio and later given the boundary conditions for the geometry and for the media. This file imported into Fluent software and has given the input values like velocity, mass flow rate, pressure, temperature etc.,

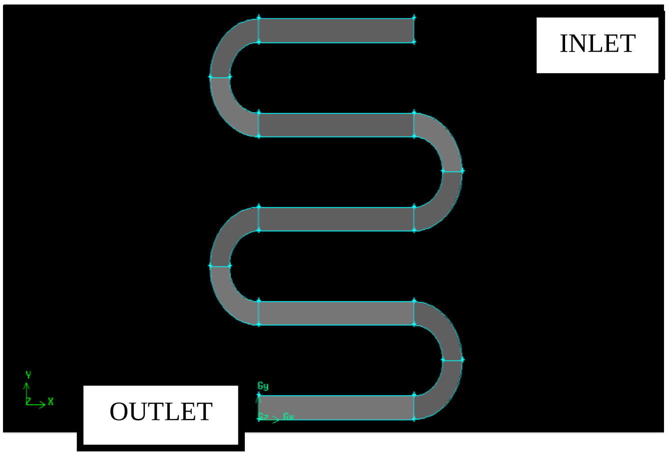

Fig.2.1 Two phase model of exchanger.

Two dimensional geometry was used to study the flow in the Heat Exchanger for solving the mass, momentum, and energy equations. The phase velocities were defined at the inlet boundary of the heat exchangers. The κ-ε turbulence models with standard wall functions were used to solve the problems. The gravitational acceleration of 9.81 m/s2 in downward flow direction was used. The simulation was done in both steady state and as well as in unsteady state, results are plotted. To the inlet of the heat exchanger mass flow rate is 0.5kg/sec. The material of pipe Al and Cu is taken and compared the values of heat transfer rate between them is more.

3. GEOMETRY DETAILS

The geometry was done in the GAMBIT with measurements, pipe diameter is 15mm, radius of the pipe 25mm and length of the pipe 80mm. Defining required boundaries like inlet, outlet and wall of the geometry and mesh under tetrahedron. Defining the boundary conditions for the coolant as the water. The figure shows the geometry of the fluid flow.

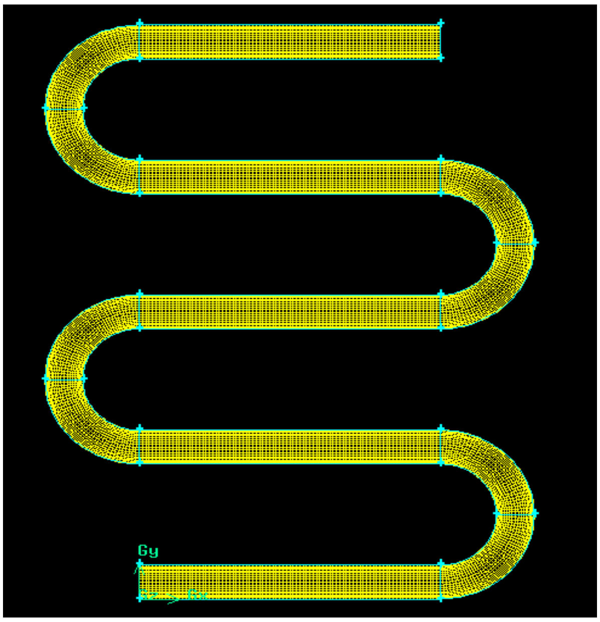

Fig.3.1 Mesh geometry of exchanger.

4. SOLUTION STRATEGY.

The simulation is done in the FLUENT based upon the governing equations. The steps followed in the fluent are define Model, define Material, define cell zone, boundary condition, solve, iterate, and analyze results. The governing equations used to solve this problem as below.

A. CONTINUITY EQUATION

Continuity Equation also called conservation of mass. The overall mass balance is

Input – output = accumulation

Assuming that there is no storage the Mass input = mass output.

However, as long as the flow is steady (time-invariant), within this tube, since, mass cannot be created or destroyed then the above equation will be

$$\dot{m}_1 = \dot{m}_1 \quad (1)$$

$$\frac{dm_1}{dt} = \frac{dm_1}{dt} \quad (2)$$

$$\rho A_1 u_1 = \rho A_2 u_2 \quad (3)$$

$$A_1 v_1 = A_2 v_2 \quad (4)$$

B. MOMENTUM EQUATION AND BERNOULLI EQUATION

It is also called equation of motion .According to Newton's 2nd law (the time rate of change of momentum of the fluid particles within this stream tube slice must equal to the forces acting on it).

F = mass* acceleration

Consider a small element of the flowing fluid as shown below, Let

dA : cross-sectional area of the fluid element,

dL : Length of the fluid element,

dW : Weight of the fluid element,

u : Velocity of the fluid element,

P : Pressure of the fluid element.

Assuming that the fluid is steady, non-viscous (the frictional losses are zero) and incompressible (the density of fluid is constant).

The forces on the cylindrical fluid element are,

Pressure force acting on the direction of flow (PdA).

Pressure force acting on the opposite direction of flow [(P+dP)dA].

A component of gravity force acting on the opposite direction of flow (dW sin θ).

Hence, Total force = gravity force + pressure force

The pressure force in the direction of low

Fp = PdA – (P+dP) dA = – dPdA (5)

The gravity force in the direction of flow

Fg = – dW sin θ {W=m g = ρ dA dL g}.

= – ρ g dA dL sin θ {sin θ = dz / dL}.

= – ρ g dA dz. (6)

The net force in the direction of flow

F = m a {m = ρ dA dL .

= ρ dA dL a.

= ρ dA u du. (7)

We have

ρ dA u du = – dP dA – ρ g dA dz {÷ ρ dA }

dP/ ρ + udu + dz g = 0 --------- Euler's equation of motion.

Bernoulli's equation could be obtain by integration the Euler's equation.

∫dP/ ρ + ∫udu + ∫dz g = constant.

P/ ρ + u2/2 + z g = constant.

ΔP/ ρ + Δu2/2 + Δz g = 0 -- Bernoulli's equation.

5. RESULTS

A. RESULTS UNDER STEADY STATE.

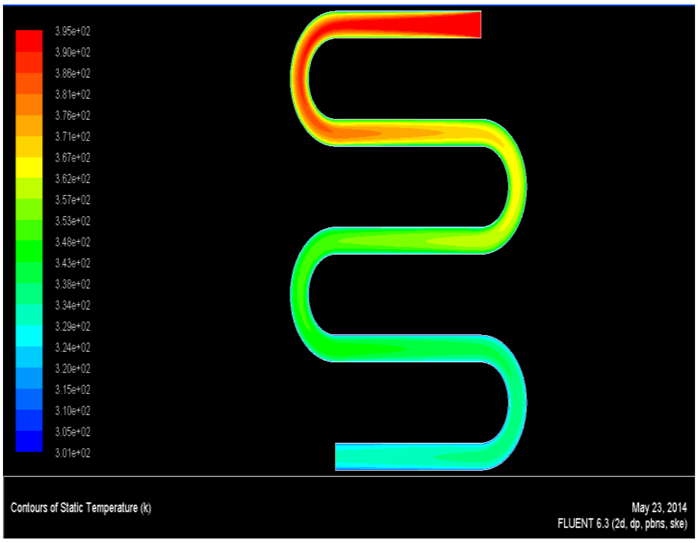

Fig.5.1 Temperature contours of exchanger.

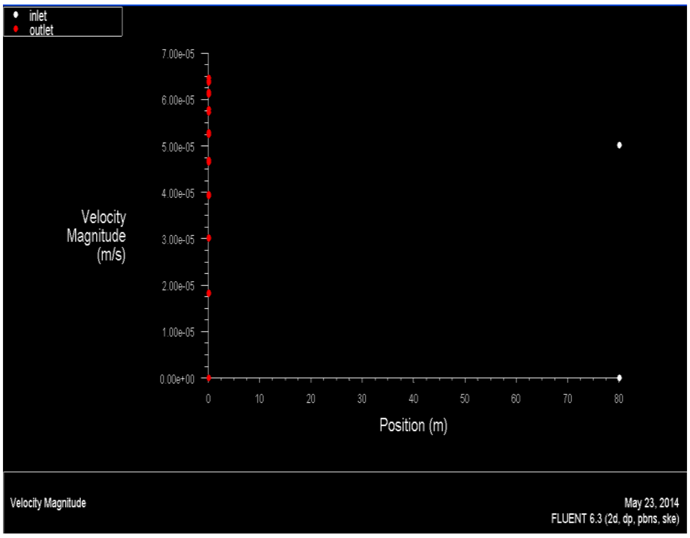

Chart-5.2 Velocity at inlet and outlet.

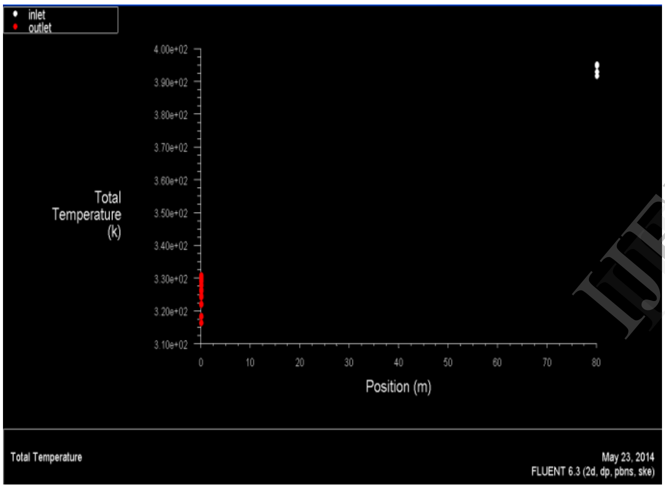

Chart-5.1 Temperature at inlet and outlet.

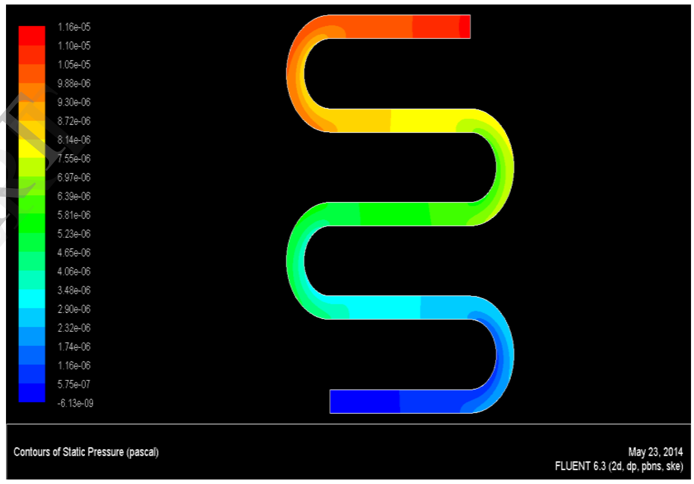

Fig.5.3 Pressure contours of exchanger.

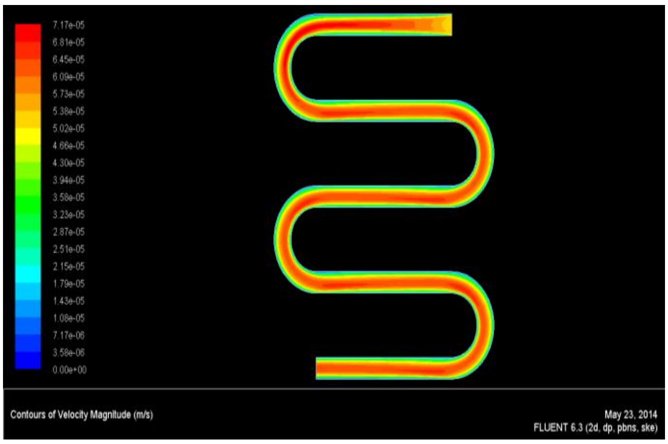

Fig.5.2 Velocity contours of exchanger.

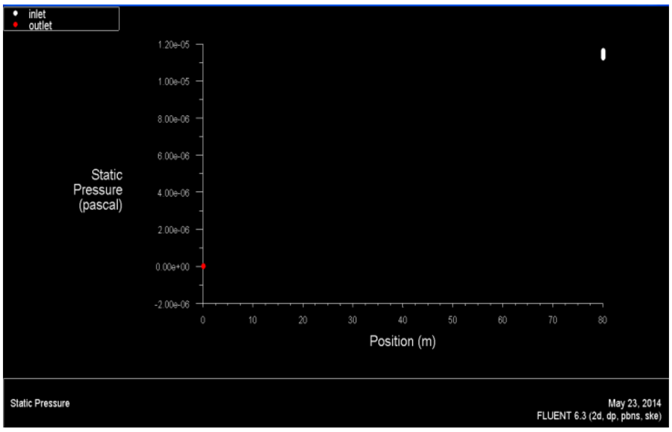

Chart-5.3 Pressure at inlet and outlet

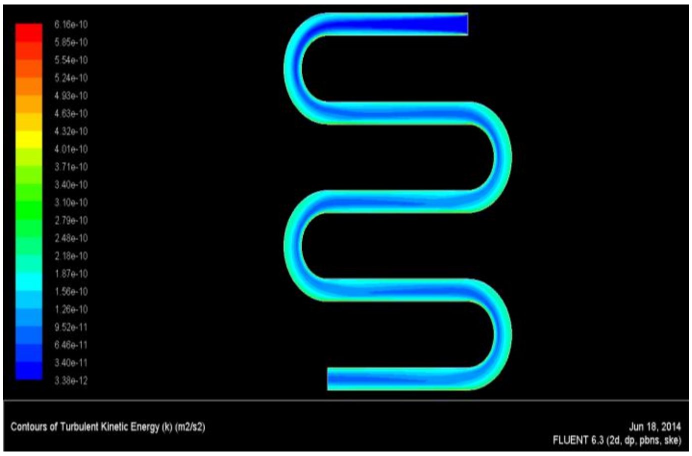

Fig.5.4 Turbulence contours of exchanger.

TABLE 5.1 RESULTS OF FLOW ANALYSIS.

s.no	parameters	Min.	Max.
1	Temperature(K)	3.01e+02	3.95e+02.
2	Velocity(m/s)	0	7.17e-05.
3	Pressure(Pascal)	-6.13e-09	1.16e-05.
4	Turbulent(pascal)	3.36e-12	6.157e-10.

B. COMPARISON OF HEAT TRANSFER RATE OF AL & CU:

TABLE 5.2 Results of Heat Transfer.

Boundary	Total heat transfer rate(W) of AL	Total heat transfer rate(W) of CU
Inlet	202542.87	202542.7
Outlet	-62895.426	-63141.576
Wall	-139642.09	-139395.92
Net	5.3509724	5.1983717

6. CONCLUSION

The flow through the radiator/heat exchanger was numerically simulated with water by steady and also by unsteady flow in k-epsilon scheme. The major observations made related to the pressure,temperature and velocity contours in the process of flow through these heat exchanger. The values(heat transfer rate) are compared with material of pipe in the heat exchagers.

Finally from the results, it is found the alliminum is having more heat transfer rate compared with copper material. The radiator with circular cross section values are plotted above.

So, the life of the heat exchanger is incresed in cases of circular cross section making the corners more round so as to minimize the losses in the pipes.

To conclude, this examination results indicate that FLUENT can be used with high degree of accuracy to visualize the temperature variations takes place in heat exchangers.

7. REFERENCES

[1] Diaper, A.D. and Hesler, L.E., (1990), "Crossflow Pressure Drop and Flow Distributions within a Tube Bundle Using Computational Fluid Dynamic", Proc. 9th Proc. 9th Heat Transfer Conf., Israel, pp. 235-240.

[2] Jian-Fei Zhang, Ya-Ling He, Wen-Quan Tao , " 3d numerical simulation of shell and tube heat exchanger with middle-overlapped helical baffle", ajournal ,School of energy and power engineering,china.

[3] Li, H., Kottke, "Effect of baffle spacing on pressure dropand local heat transfer in shell and tube heat exchangers for staggered tube arrangement", source book on Int.J. Heat Mass Transfer 41 (1998), 10, pp. 1303–1311.

[4] Thirumarimurugan, M., Kannadasan, T., Ramasamy, E., Performance Analysis of Shell and Tube Heat Exchanger Using Miscible System, American Journal of Applied Sciences 5 (2008), pp. 548-552.

[5] Usman Ur Ehman , G¨oteborg, Sweden 2011, Master's Thesis 2011:09 on "Heat Transfer Optimization of Shell-and-Tube & Heat Exchanger through CFD".

[6] Colebrook C.F.,"Turbulent Flow in Pipes with Particular reference to the Transition Region between the Smooth and Rough Pipe Laws", J. Inst. of Civil Engrs.N° 11,(1939), pp.133-156.

[7] Bhave, P.R.,"Analysis of flow in water distribution networks", pp.25-68 Technomic Pub. Co., Inc., USA, (1991).

[8] Rouse H., "Evaluation of Boundary Rough Proc.2ndHydraulic. Conf., Bulletin N° 27, Univ. of Lowa, Lowa City, (1943).

[9] Moody L.F., "Friction Factors for Pipe Flow", Trans. American Society of Mechanical. Engineers. No. 66, (1944), pp. 671-684.

[10] Hazen-Williams Formula http://www.pipeflow.com/pipe-pressure-drop-calculations/ pipe-friction-loss .

[11] Ansys,Inc.http://http://www.idac.co.uk/products/downloads/ Meshing.pdf

[12] Tobias Zitzmann1, Malcolm Cook2, Peter Pfrommer1, Simon Rees2, Ljiljana Marjanovic2, "Simulation Of Steady-State Natural Convection Using CFD" Ninth International IBPSA Conference Montréal, Canada.