International Research Journal in Global Engineering and Sciences. (IRJGES)
ISSN : 2456-172X | Vol. 3, No. 1, March - May, 2018
Pages 140-147 | Cosmos Impact Factor (Germany): 5.195
Received: 02.04.2018 Published : 24.04.2018

NUMERICAL SIMULATION TO REDUCE THE SCALE FORMATION IN SHELL AND TUBE HEAT EXCHANGER

S.Yuvaraj[1], I.Satham hussain[2], M.Gurumaharaj[3]

[1,2,]Assistant professor, Department of Mechanical Engineering,
Dhanalakshmi Srinivasan College of Engineering,Coimbatore – 641 105.
[3,]Associate professor, Department of Mechanical Engineering,
Dhanalakshmi Srinivasan College of Engineering Coimbatore – 641 105.

ABSTRACT

In the contemporary era it is mandatory to use of shell and tube heat exchanger in the industrial and power plant applications. A deposit of fouling reduces the heat transfer in the heat exchangers. The current paper studies about used copper rings to reduce scale formation in the shell side of shell and tube heat exchangers. There are three different rings geometry were studied numerically to improve the heat transfer rate. CFD software was used for the numerical simulation. The simulated results shows that conical shape rings produces better heat transfer enhancement of 317.67K.

I. INTRODUCTION

There are many reasons are identified for reducing the efficiency of heat transfer. Fouling formation plays a vital role to increase or decrease the heat transfer rate. Deposits of scales and unwanted dirt in the inner and outer surface of heat exchangers, so the surface of the heat exchanger is increased and it creates the resistance in a heat exchanger [1]. Shell and tube heat exchangers are designed and manufactured based on the local temperature and the velocity of the fluid which is used for heat transfer from that temperature is the main phenomena of fouling in industrial heat exchangers [2]. Fluidized bed heat exchanger is one of the type of heat exchanger in this fluidized bed systems Particle hitting on the wall, particle- particle collision are the two methods for prevent and reduce the scale formation[3]. Formation of fouling was monitored and it is reduced when we give the hydraulic shock at the time of fluid flow[4]. Chinese Professor, Su Jingxin investigated about fouling corrosion in aluminium heat exchangers and he found corrosive behavior and weight is reduced with a covered layer of artificial fouling in a humid atmosphere [5].This paper investigates the influence of copper grooves in shell and tube heat exchanger based on measurements on two different types of copper grooves (plane edge and sharp edge).

International Research Journal in Global Engineering and Sciences. (IRJGES)
ISSN : 2456-172X | Vol. 3, No. 1, March - May, 2018
Pages 140-147 | Cosmos Impact Factor (Germany): 5.195
Received: 02.04.2018 Published : 24.04.2018

II. GEOMETRY MODELLING AND MESHING

For CFD simulation the geometry of the shell and tube heat exchanger is created using 'SOLID WORKS'. After creating the geometry, extracting the fluid region is the next step in which all the surfaces which are in the contact of fluid are taken alone and all other surfaces are created. This clean-up is done in ANSA meshing tool which is very robust clean up tool. After cleaning up the geometry surface mesh is generated in ANSA tool itself. All the surfaces are discredited using tri surface element .As the geometry has some complicated and skewed surfaces tri surface elements are used to capture the geometry. Volume mesh is generated in T-Grid which is a robust volume mesh generator. Volume is discretised using tetrahedron .Each and every cell centroid is the co-ordinate at which the Navier-stokes systems of equations are solved. ANSYS-FLUENT is used as the solver. Here the fluid flow is assumed to be three dimensional and turbulent. After selection of turbulence model boundary conditions are specified. Fluent has capability to store value of physical parameters for any point in the domain for analysis. Seven points is created to store the value of physical parameters such as temperature, velocity, and pressure. FLUENT is used to simulate flow problem and finally post processing was done in the results.

The generated model is exported to the further process in the form of IGES as it is a third party format which can be taken into any other tools. Here, Two type of copper coil is used (sharp and flat edge) to create some kind of localized suction in between the copper coils due to the condensation process. So, it avoids the scale or fouling formation. The Sharp edge copper coil is more efficient when compared with the Flat edge copper coil. After making the geometry of the domain, next step is to mesh the domain. The CFD tool was used to create the fine mesh quality. When considering case-1, case-2 (sharp grooves) and case-3 (plain grooves), the surface and volume mesh is generated with 5.25 and 18.89 lakhs, and 18.96 lakhs, 4.27 and 17.24 lakhs respectively. This mesh contains tetrahedral cells having triangular faces at the boundaries are shown in fig-1,2,3,.The mesh details are given in Table 1.

Table 1. Mesh details of three cases

Model	Surface mesh	Quality	Volume mesh	Quality
Base case	525670	0.6	1889715	0.8499
Mod-1 (sharp grooves)	509450	0.6	1896024	0.8599
Mod-2 (plain grooves)	427000	0.6	1724411	0.9324

Steel tube shell and tube heat exchanger is taken as base case model of shell and tube heat exchanger without any copper wounding which having 3mm thickness and 3m

International Research Journal in Global Engineering and Sciences. (IRJGES)
ISSN : 2456-172X | Vol. 3, No. 1, March - May, 2018
Pages 140-147 | Cosmos Impact Factor (Germany): 5.195
Received: 02.04.2018 Published : 24.04.2018

length. In this case the outlet temperature of the fluid is 321.09K. This case of heat exchanger having more scale formation of 0.3mm and less amount of heat transfer rate. Other two types of heat exchangers are having copper wounding of plane and sharp edges as shown in Fig. 1 and 2 respectively.

Once the mesh generation is completed boundary condition are defined for CFD domain. Specify “boundary condition” icon is used to create boundaries. In FLUENT launcher, both fluid and solid can be defined. Generally, the copper materials used in this analysis. The fluid used in this analysis is water vapour. The material and fluid properties are mentioned in Table 2.

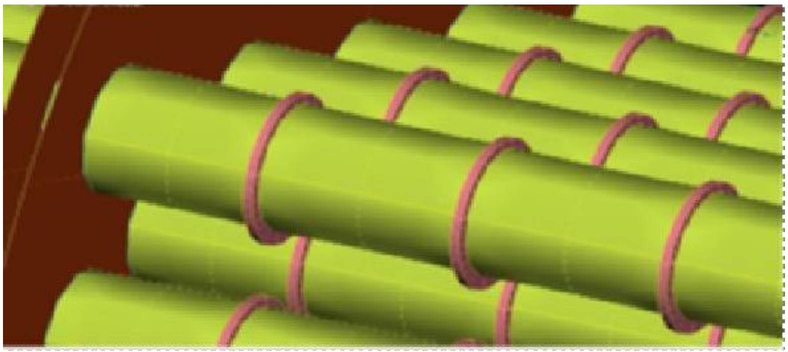

Fig.1. Heat exchanger with plane grooves

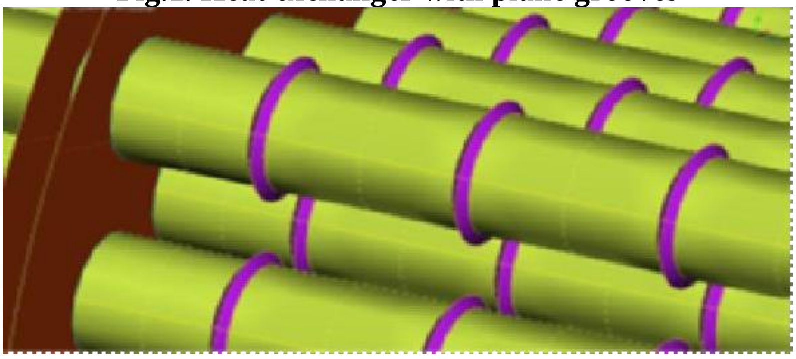

Fig.2. Heat exchanger with sharp grooves Table 2.Material and fluid properties

Table 2 Material and fluid properties

Fluids Solids	Properties		
	Density kg/m^3	Specific heat J/kg-K	Thermal conductivity W/mK
Steam	0.598	1858.68	0.0261
Water	998.2	4182	0.6
Steel	8030	502.48	16.27
Copper	8978	381	387.6

International Research Journal in Global Engineering and Sciences. (IRJGES)
ISSN : 2456-172X | Vol. 3, No. 1, March - May, 2018
Pages 140-147 | Cosmos Impact Factor (Germany): 5.195
Received: 02.04.2018 Published : 24.04.2018

III. GOVERNING EQUATIONS

The basic governing equations, which describe the fluid dynamics, are used to solve the steam and water flow. The energy equation was used to define the conductive heat transfer across the fluid through the solid region.

Conservation of Mass

$$\partial\rho/\partial\tau + \mathrm{div}\,(\rho u)=0 \qquad (1)$$

Conservation of X Momentum

(qu/)&v+ *div* (quu)=− &(p)/&x+ *div* (μgradu)+ *SMx* (2)

Conservation of Y Momentum

(qv)&v+ *div* (qvu)=− &(p)&y+ *div* (μgradv)+ *SMy* (3)

Conservation of Z Momentum

$$\partial(\rho w)\partial\tau+\mathrm{div}\,(\rho wu)=-\,\partial(p)\partial z+\mathrm{div}\,(\mu\ \mathrm{grad}\ w)+SMz \qquad (4)$$

IV RESULTS AND DISCUSSION

By completion of all the test runs in Fluent, The rate of heat transfer characteristics in shell and tube heat exchanger for 3 different cases are reported. For the all there cases the inlet temperature of the steam is 373K. In the base case model having the normal heat exchanger pipe which was not having any wounding and it gives outlet temperature of 321.09K. Other two cases result shows that there is an improvement in turbulent intensity of hot and cold fluid mixing. Turbulence intensity increases because of wounding the copper rings. So, there is an enhancement in heat transfer.

Comparison of three different cases

The rate of heat transfer depends upon the turbulence intensity, different cases [Base model, Modification 1 (sharp groove), Modification 2 (plain edge)] of rate of heat transfer was analysed in CFD simulation software. The details are mentioned in Table 2.

In sharp edge grooves result shows that the outlet temperature of the steam is 317.67K. Compared to the base case model this is the lower temperature of steam outlet and higher turbulence intensity because of sharp edge copper grooves. This sharp edge creates more turbulence intensity at the corner's of the rings and velocity of the fluid is varied at the place

International Research Journal in Global Engineering and Sciences. (IRJGES)
ISSN : 2456-172X | Vol. 3, No. 1, March - May, 2018
Pages 140-147 | Cosmos Impact Factor (Germany): 5.195
Received: 02.04.2018 Published : 24.04.2018

of copper wounding. Copper wounding creates some amount of emf. So small amount of current is produced and it did not affect the velocity of the fluid.

The Fig 3 shows the temperature of heat exchanger is increased in the sharp edge grooves. So outlet temperature of the fluid is 317.67K because of sharp edge grooves. This copper rings having 3.55mm thickness, 4mm length and the sharp edge angle is 45° which is fixed in 54 mm pitch. Copper having thermal conductivity so it absorbs the temperature easily, because of high temperature and high turbulence intensity of hot and cold fluid. The turbulence intensity variations of sharp edges are shown in Fig 5.

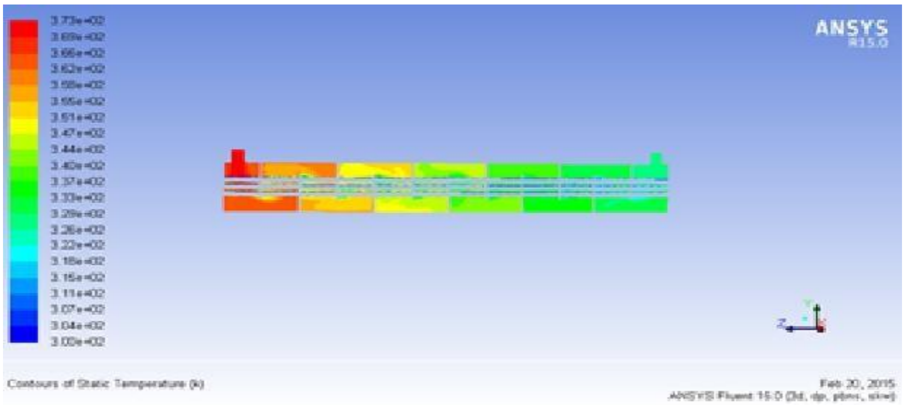

Fig.3. Static temperature for sharp grooves

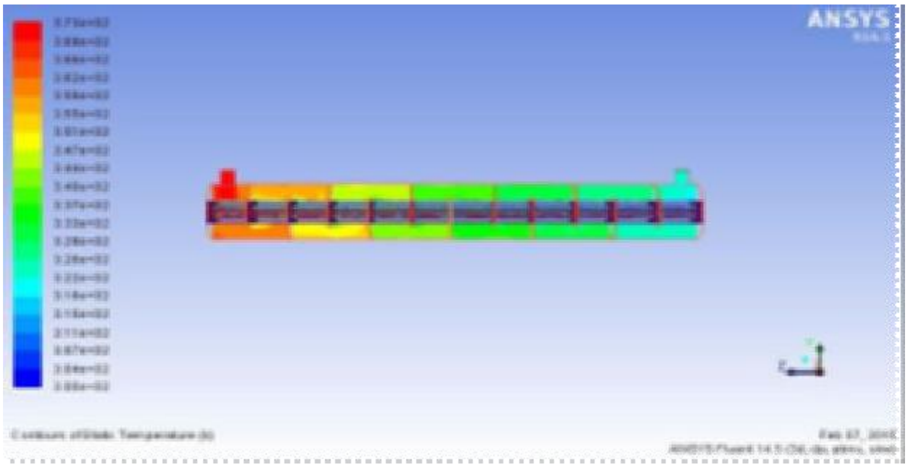

Fig.4. Static temperature of plane grooves

In plane edge copper grooves the turbulence intensity is higher than the base case and lower than the sharp edge grooves. 2.5mm length 5mm thickness plane grooves are used for this analysis. These grooves are fixed in the same distance 54mm. outlet temperature of the plane edge grooves heat exchanger is 319.41K. This is lower than the sharp edge corner

International Research Journal in Global Engineering and Sciences. (IRJGES)
ISSN : 2456-172X | Vol. 3, No. 1, March - May, 2018
Pages 140-147 | Cosmos Impact Factor (Germany): 5.195
Received: 02.04.2018 Published : 24.04.2018

heat exchanger. Due to producing of emf, the intensity of the fluid is also varied. The Fig 4 and Fig 6 describe the temperature and turbulence variation of plane edge grooves heat exchanger.

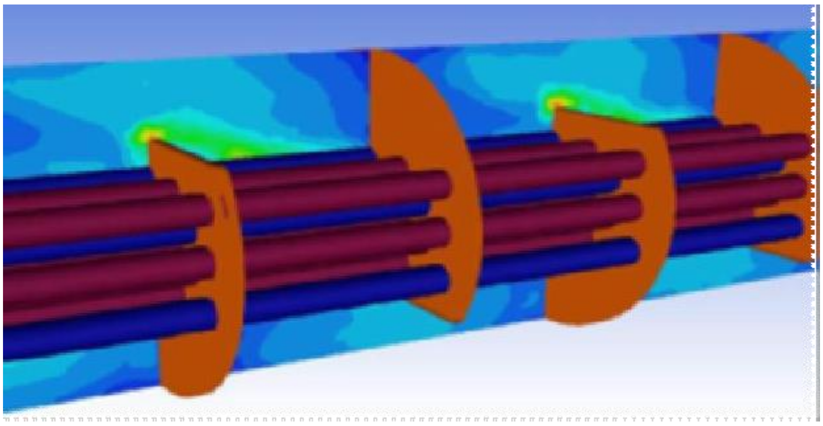

Fig.5. Turbulence intensity of sharp grooves

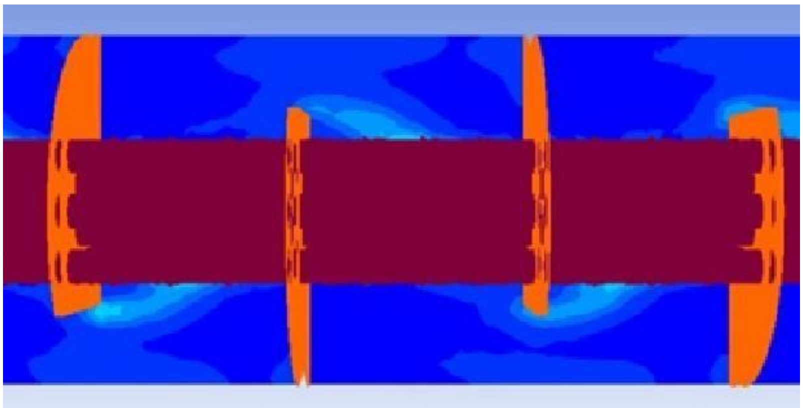

Fig.6. Turbulence intensity of plane groves

Fluid velocity plays a vital role in the heat transfer in the shell and tube heat exchanger. Steel tube diameter of 88mm and inlet valve diameter of 80 mm. Fluid flows over the heat exchanger whose horizontal distance between the tubes is 80 mm, vertical distance between the tubes are 60 mm and the inclined angle between two tubes are 45°. So the fluid is flows over the tube easily. The produced emf did not affect the fluid flow at any situation, but it affects the scale formation. If the flow of fluid is high means the scale formation is also high, flow rate of fluid id no means the scale formation is also low. The various velocity magnitudes are shown in the Fig 7 and 8. Plane heat exchanger pipe creates 0.3 mm thickness of scale forming. Instead of that pane pipe pane and sharp edge copper grooves are used to reduce scale formation and increase the rate of heat transfer. In the sharp edge grooves type only 0.1-0.15 mm thickness scale only formed. This is very lowest scale formation compare to other cases. Thickness of the plane edge copper grooves is 0.25mm.

International Research Journal in Global Engineering and Sciences. (IRJGES)
ISSN : 2456-172X | Vol. 4, No. 3, September - November, 2019
Pages 1-10 | Cosmos Impact Factor (Germany): 5.195
Received: 02.09.2019 Published : 24.09.2019

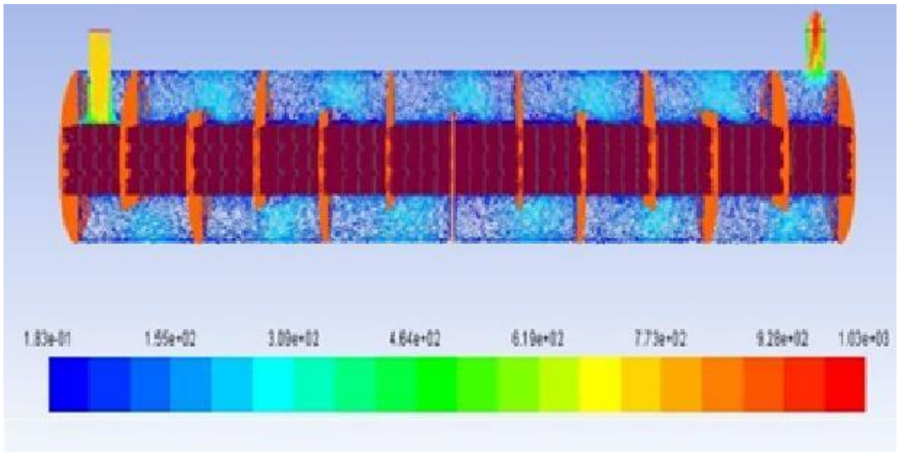

Fig.7. Contours of velocity magnitude (sharp grooves)

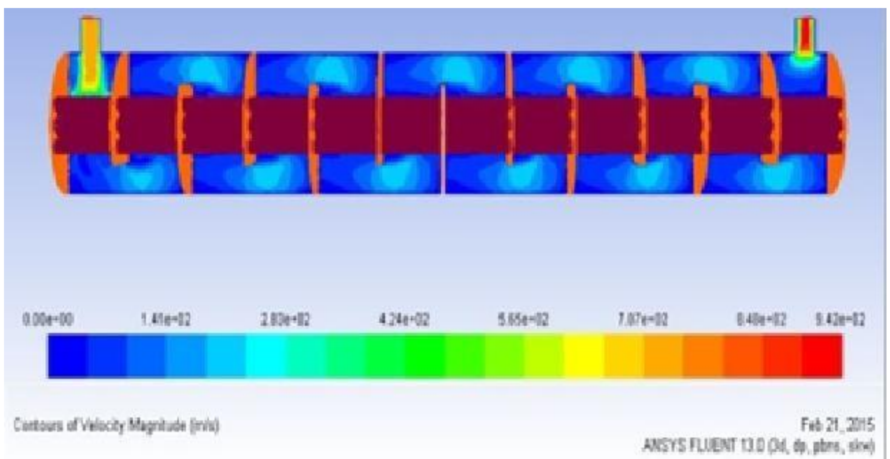

Fig.8. Contours of velocity magnitude (plane grooves)

V CONCLUSION

This novel approach is taken to enhance the condensation heat transfer rate in a shell and tube heat exchanger by introducing plain and sharp grooves CFD is utilized to understand and analyze the hydro dynamic and thermo dynamic behavior of these modifications very clearly. It was found that the sharp grooves increases the local turbulence intensity, suction, momentum and heat transfer much better than other two models consequently has higher heat transfer rate over other two modifications. The outlet temperature of the sharp edge grooves is 317.67K and thickness of the scale in sharp edge grooves is 0.1mm. So, In industrial used applications and other type of applications sharp edge copper wounding heat exchangers are best for increasing the rate of heat transfer.

International Research Journal in Global Engineering and Sciences. (IRJGES)
ISSN : 2456-172X | Vol. 4, No. 2, June - August, 2019
Pages 1-10 | Cosmos Impact Factor (Germany): 5.195
Received: 02.09.2019 Published : 24.09.2019

VI. REFERENCES

1. S.B. Genic, Experimental determination of fouling factor on plate heat exchangers in district heating system, Energy and Buildings, Vol.50, 2012, pp.204- 211.
2. D. Butterworth, Design of shell and tube heat exchangers when the fouling depends on local temperature and velocity, Applied Thermal Engineering, Vol.22, 2002, pp.789-801.
3. P. Prank, Prevention of fouling and scaling in stationary and circulating liquid solid fluidized bed heat exchangers: Particle impact measurements and analysis, International Journal of Heat and mass Transfer, Vol.52, 2009, pp. 3857-3868.
4. N. Senthilkumar, C.K. Dhinakarraj, B. Deepanraj, N.M. Babu, A. Santhoshkumar, Modification and analysis of compressor intercooler fin in turbocharger using FEM, Procedia Engineering, Vol.38, 2012, pp.379-384.
5. M.A.S. Jeronimo, Monitoring the Thermal Efficiency of Fouled Heat exchangers: A Simplified Method, Experimental Thermal and Fluid Science, Vol.14, 1997, pp. 455-463.
6. S. Jingxin, Fouling Corrosion in aluminium heat exchangers, Chinese Journal of Aeronautics, Vol.28, 2015, pp. 954-960.
7. B. Deepanraj, P. Lawrence, G. Sankaranarayanan, Theoretical analysis of gas turbine blade by finite element method, Scientific World, Vol.9, 2011, pp.29- 33.