“optimization of heat transfer coefficient of sthe using CFD”

DILIP S PATEL [1], RAVINDRASINH PARMAR [2], VIPUL M PRAJAPATI[3]

[1] Associate Prof., Department of Mechanical Engineering S.K.Patel college of Engg. Visnagar, Gujarat, India
[2] Student of ME, Department of Mechanical Engineering S.K.Patel college of Engg. Visnagar, Gujarat, India
[3] Assistant Prof., Department of Mechanical Engineering S.K.Patel college of Engg. Visnagar, Gujarat, India

---***---

Abstract - *Computational Fluid Dynamics(CFD) can very useful to gain visualize the flow and temperature fields on the shell side can simplify the assessment of the weakness. In this present study ,attempts were made to investigate the impacts of various mass flow rates on fluid flow and the heat transfer characteristics of a shell-and-tube heat exchanger for 0° and 20° baffle inclination angle. The simulation results for different angle are compared for their performance. The study is concerned with a single shell and single side pass parallel flow heat exchanger. The flow and temperature fields inside the shell are studied using non-commercial computational fluid dynamics software tool ANSYS CFX 14.5.From the computational fluid dynamics simulation results, the shell side outlet temperature, pressure drop, recirculation near the baffles, optimal mass flow rate and heat transfer graph are determined for the given heat exchanger geometry.*

Key Words: *Shell and tube heat exchanger, CFD, turbulence model, computational modeling, Fluent.*

1.INTRODUCTION

Shell and tube heat exchangers consist of a bundle of parallel tubes that provide the heat transfer surface separating the two fluid streams. The tube side fluid passes axially through the inside of the tubes. The shell side fluid passes over the outside of the tubes. The process fluid is usually placed inside the tubes for ease of cleaning or to take advantage of the higher pressure capability inside the tubes. The thermal performance of such an exchanger usually surpasses a coli type but is less than a plate type. Pressure capability of shell and tube heat exchanger is generally higher than a plate type.

This heat exchanger shown in Fig.1 is generally built of a round tubes mounted in a cylindrical shell with the tube axis parallel to that of the shell. One fluid flows inside the tubes, the other flows across and along the tubes. The major components of this exchanger are tubes,shell,and front-end head, rear-end head, baffles and tube sheets.

A variety of different internal constructions are used in shell-and -tube exchangers, depending on the desired heat transfer and pressure drop performance and the methods employed to reduce thermal stresses, to prevent leakages.

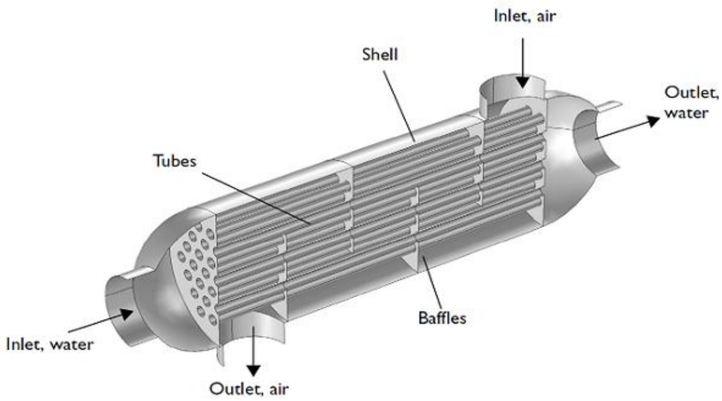


Fig.1 : Shell and tube heat exchangers

1.1 COMPUTATIONAL MODELING

A computational model is a mathematical model in computational science that requires extensive computational resources to study the behavior of a complex system by computer simulation. The system under study is often a complex nonlinear system for which simple ,intuitive analytical solutions are not readily available.

Rather than deriving a mathematical analytical solution to the problem, experimentation with the model is done by adjusting the parameters of the system in the computer, and studying the differences in the outcome of the experiments. Operation theories of the model can be derived/deduced from these computational experiments. The computational modeling involves pre-processing, solving and post-processing.

1.2 Geometry modeling

The cavity model is designed according to TEMA(Tubular Exchanger Manufacturers Association) Standards Gaddis (2007),using Creo software. Design parameters and fixed geometric parameters have been taken similar to Ender Ozden et al.[9],as indicated in table.1.

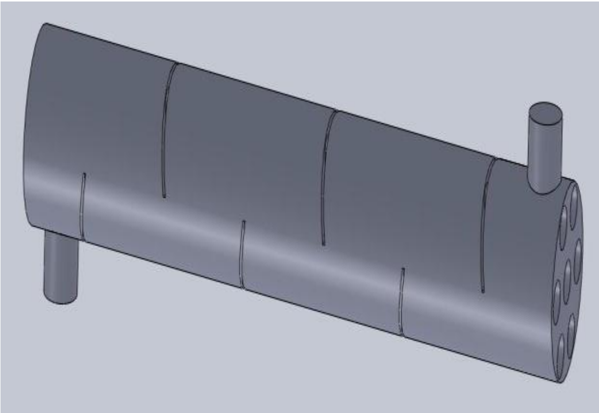


Fig-2: Cavity Model of Shell and Tube Type Heat Exchanger at 0° baffle inclination

Table-1: Design parameters of shell and tube heat exchanger

Heat exchanger length	3000 mm
Shell inner diameter	135 mm
Tube outer diameter	9 mm
Number of tubes	28
Number of baffles	6
Central baffle spacing	500 mm
Baffle inclination angle	0°
Tube Material	Carbon steel

2. . Mesh Generation

The partial differential equation that governs fluid flow and heat transfer are not usually amenable to analytical solutions, except for very simple cases. Therefore, in order to analyze fluid flows, flow domains are split into smaller sub domains (made up of geometric primitives like hexahedra and tetrahedral in 3D, and quadrilaterals and triangles in 2D) and discredited governing equations are solved inside each of these portions of the domain. Mesh generation is the practice of generating a polygonal or polyhedral mesh that approximates a geometric domain.

The entire geometry is divided into three fluid domains Fluid Inlet, Fluid Shell and Fluid Outlet.

Details of Meshing

Type of Analysis: - 3D

Type of Element: - Tetrahedral (10 Node)

Physical preference-CFD

Solver preference-CFX

Use advance size of function- On Curvature

Relevance centre- Coarse

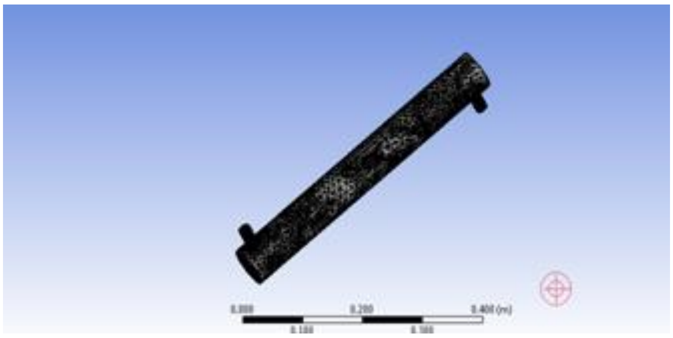


Fig-3: Meshing of Heat Exchanger

Governing equations

The governing equations of the flow are modified according to the conditions of the simulated case. Since the problem is assumed to be steady, time dependent parameters are dropped from the equations are:

Conservation of mass: $\nabla(\rho V_r) = 0$

x-momentum :

$$\nabla.(\rho u V_r) = -\frac{\partial p}{\partial x} + \frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z}$$

y-momentum :

$$\nabla.(\rho v V_r) = -\frac{\partial p}{\partial y} + \frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \tau_{yy}}{\partial y} + \frac{\partial \tau_{zy}}{\partial z} + \rho g$$

z-momentum :

$$\nabla.(\rho w V_r) = -\frac{\partial p}{\partial z} + \frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zz}}{\partial z} + \rho g$$

Energy: $\nabla(\rho e V_r) = -p \nabla V_r + \nabla(k \nabla T) + q_\varphi$ ----(1)

In Eq.(1), φ is the dissipation function that can be calculated from

$$\varphi = \mu[2[\left(\frac{\partial u}{\partial x}\right)^2 + \left(\frac{\partial v}{\partial y}\right)^2 + \left(\frac{\partial w}{\partial z}\right)^2] + \left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\right)^2 + \left(\frac{\partial u}{\partial z} + \frac{\partial w}{\partial z}\right)^2 + \left(\frac{\partial v}{\partial z} + \frac{\partial w}{\partial y}\right)^2] + \lambda(\nabla.V_r)^2$$

C. Turbulence Model

The realizable k-ε model is a relatively recent development and differs from the standard k-ε model in two important ways:

- The realizable k-ε model contains a new formulation for the turbulent viscosity.
- A new transport equation for the dissipation rate, ε, has been derived from an exact equation for the transport of the mean-square vortices fluctuation.

Transport equations for the Realizable k-ε model

The modeled transport equations for k and ε in the realizable k-ε model are

$$\frac{\partial}{\partial t}\rho k + \frac{\partial}{\partial x_j}(\rho k u_j) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu t}{\sigma_k}\right)\frac{\partial k}{\partial x_j} + G_k + G_b - \rho\varepsilon - Y_M + S_k\right.$$

and

$$\frac{\partial}{\partial t}(\rho\varepsilon) + \frac{\partial}{\partial x_j}(\rho\varepsilon u_j) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu t}{\sigma_\varepsilon}\right)\frac{\partial\varepsilon}{\partial x_j}\right] + \rho C_1 S_\varepsilon - \rho C_2\frac{\varepsilon^2}{k+\sqrt{v\varepsilon}} + C_{1\varepsilon}\frac{\varepsilon}{k}C_{3\varepsilon}G_b + S_\varepsilon$$

where $C_1 = Max\left\{\left[0.43, \frac{n}{n+5}\right], n = \frac{Sk}{\varepsilon}\right.$, $s=\sqrt{2S_{ij}S_{ij}}$

In these equations, G_k represents the generation of turbulence kinetic energy due to the mean velocity gradients, G_b is the generation of turbulence kinetic energy due to buoyancy, Y_M represents the contribution of the fluctuating dilatation in compressible turbulence to the overall dissipation rate.

C_2 and $C_{1\varepsilon}$ are constants., σ_k and σ_ε are the turbulent Prandtl numbers for **k** and **ε**,respectively. S_k and S_ε are user-defined source terms.

Boundary conditions:-

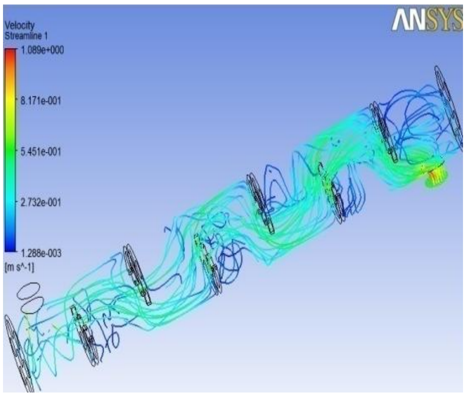
1. In the shell side working fluid is water,
2. The inlet temperature of water is set to 300 ºC,
3. The shall temperature constant at 450 ºC is assigned to the tube walls,
4. At the outlet nozzle Zero gauge pressure is assigned,
5. At inlet velocity profile is assumed to be uniform,
6. No slip condition is assigned to all surfaces,
7. At shell outer wall the zero heat flux boundary condition is assigned (excluding the baffle shell Interfaces), assuming the shell is perfectly insulated.

3.RESULT AND DISCUSSION

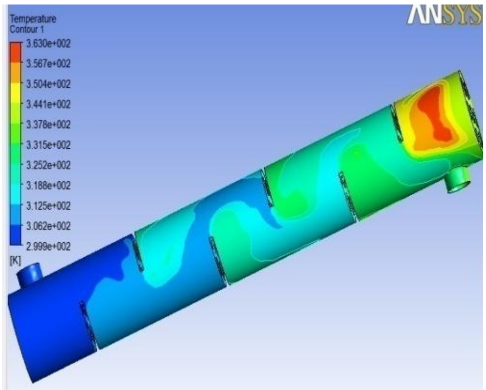
A. Validation

Simulation results are obtained for different angle. It is seen that the temperature gradually increases from 300k at the inlet to 340 k at the outlet of the shell side. The average temperature at the outlet surface is nearly 326.22 k for this model. The pressure drop is less for 2kg/s mass flow rate compared to other two mass flow rates. The maximum velocity is 3.02841 m/s for mass flow rate at the inlet and exit surface and velocity magnitude reduces to zero at the baffle surface. It can be compared that 2 kg/s mass flow gives more heat transfer than other two mass flow rates.

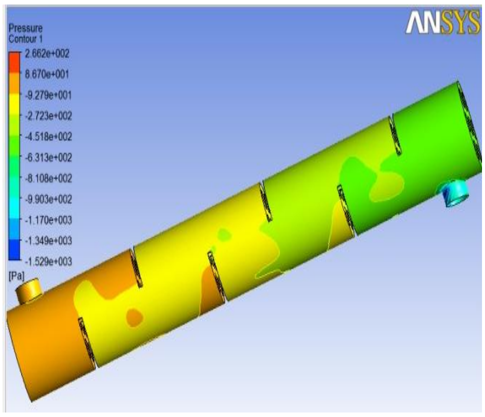
From the stream line contour, it is found that recirculation near the baffles induces turbulence eddies which would result in more pressure drop for this model. From the result table it is found that the shell outlet temperature decreases with increasing mass flow rates. Increment in mass flow rate gives also increment in velocity.



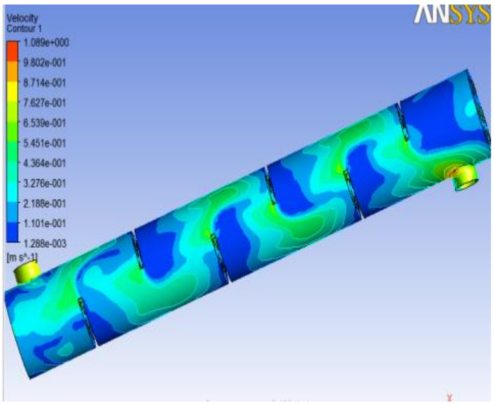
Velocity Streamline Contour



Temperature Contour

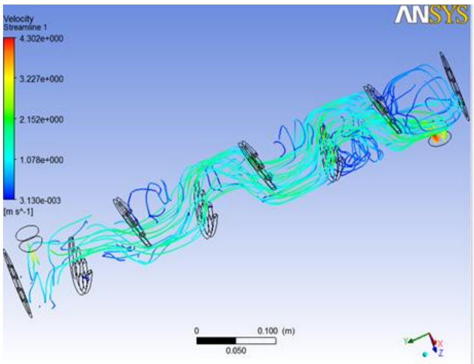


Pressure Contour

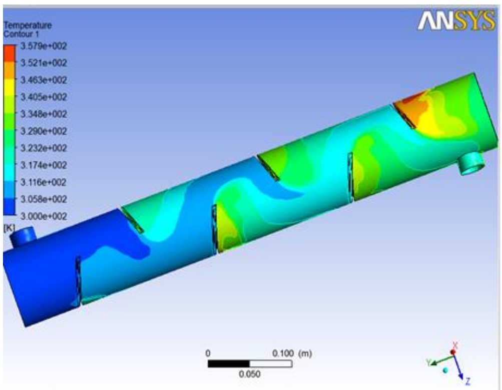


Velocity Contour

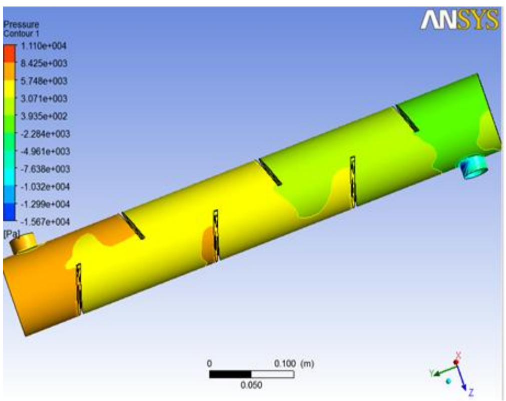
Fig.4:Velocity Streamline ,Temperature, Pressure and Velocity Contour for 0 baffle angle



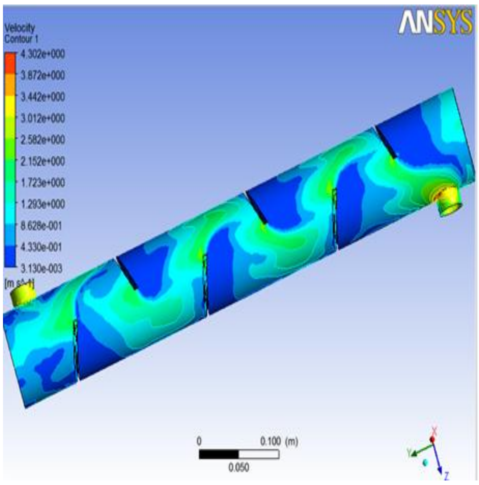
Velocity Streamline Contour



Temperature Contour

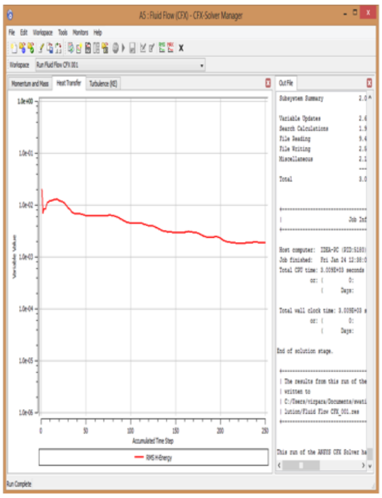


Pressure Contour



Velocity Contour

Fig.5:Velocity Streamline, Temperature, Pressure and Velocity Contour for 20 baffle angle



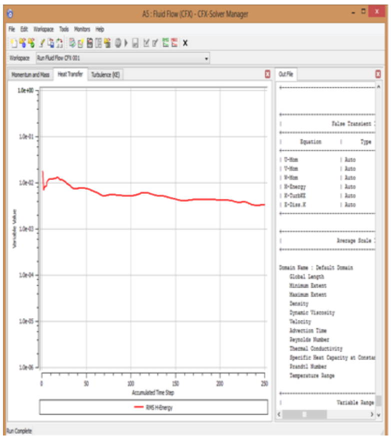


Fig.6: 0 angle baffle vs 20 angle baffle heat transfer

Table-2: CFD Result

Baffle angle	Mass flow rate[kg/s]	Outlet Temperature [K]	Velocity [m/s]
0	**2**	**330.32**	**0.7551**
20	**2**	**331.37**	**1.51**

3. CONCLUSIONS

Shell and tube heat exchanger for shell side study of the fluid flow with zero baffle inclination angle is modeled in the Pro-E Software. The initial CFD analysis have been performed in the ANSYS Software. The k-ε turbulence model is used for the simulation based on the literature survey. The initial simulation results agree with the fundamental physics of the heat transfer in the heat exchanger. The results shows the following points.

- CFD has emerged as a cost effective alternative and it provides speedy solution to heat exchanger design and optimization.
- The k - ε turbulence model has been most widely employed for heat exchanger design optimization
- From the CFD analysis it is found that the shell outlet temperature decreases with increasing mass flow rates.
- Increment in mass flow rate gives also increment in velocity.
- 20º baffle angle give good heat transfer than other angle.

REFERENCES

1. Muhammad Mahmood Aslam Bhutta, Nasir Hayat, Muhammad Hassan Bashir, Ahmer Rais Khan, Kanwar Naveed Ahmad, Sarfaraz Khan5, CFD Applications In Various Heat Exchangers Design: A Review, Department Of Mechanical Engineering, University Of Engineering & Technology, Applied Thermal Engineering, 2011.

2. Qiuwang Wang, Qiuyang Chen, Guidong Chen, Min Zeng,"Numerical investigation on combined multiple shell-pass shell-and-tube heat exchanger with continuous helical baffles",International Journal of Heat and Mass Transfer 52 (2009).

3. K.Mohammadi, W.Heidemann, H.Muller – Steinhagen "Numerical Investigation Of The Effect Of Baffel Orientation And Baffle Cut On Heat Transfer And Pressure Drop Of A Shall And Tube Heat Exchanger" ResearchGate , august 2006.

4. Hari Haran , Ravindra Reddy and Sreehari " Thermal Analysis of Shall and Tube Heat Exchanger using C and Ansys" International Journal of Computer Trends and Technology- volume 4 issue 7 –July 2013.

5. Huadong Li, Volker Kottke ," Effect Of Baffle Spacing On Pressure Drop And Local Heat Transfer In Shell-And-Tube Heat Exchangers For Staggered Tube Arrangement", International Journal of Heat Mass Transfer, Elsevier Science , Germany, 1998.

6. Ender Ozden, Ilker Tari ,"Shell Side CFD Analysis of A Small Shell And Tube Heat Exchanger" ,Middle East Technical University, 2010.

7. Apu Roy, D.H.Das, CFD Analysis Of A Shell And Finned Tube Heat Exchanger For Waste Heat Recovery Applications, National Institute Of Technology, 2011.

8. D.P. Naik and V.K. Matawala, "An assessment of counter flow shell and tube heat exchanger by entropy generation minimization method", World Journal of Science Technology, Vol. No. 2, pp. 28-32, 2012.

9. Hamidou Benzenine, Rachid Saim , Said Abboudi and Omar Imine, "Numerical analysis of a turbulent flow in a channel provided with transversal waved baffles",International Journal of thermal science, Vol. No. 17, pp. 1491-1499, 2013.

10. K.M. Lunsford, "Increasing Heat Exchanger Performance", Bryan Research and Engineering, Inc., Vol. No.1, pp. 1-13, March 1998.

11. A. E. Zohir, "Heat Transfer Characteristics in a Heat Exchanger for Turbulent Pulsating Water Flow with Different Amplitudes", Journal of

American Science, Vol. No. 8, pp. 241-250, 2012.

12. Simin Wang, Jian Wen and Yanzhong Li, "An experimental investigation of heat transfer enhancement for a shell-and-tube heat exchanger", Applied Thermal Engineering, Vol. No. 29, pp. 2433-2438, 2009.

13. J.S. Jayakumara, S.M. Mahajania, J.C. Mandala, Kannan N. Iyer, P.K. Vijayan, "CFD analysis of single-phase flows inside helically coiled tubes" ,Computers and Chemical Engineering 34 (2010).