

Optimization of Shell & Tube Heat Exchanger by Baffle Inclination & Baffle Cut

Joemer.C.S ¹, Sijo Thomas ², Rakesh.D ³, Nidheesh.P ⁴

B.Tech Student, Dept. of Mechanical Engineering, Toc H Institute of Science and Technology, Ernakulam, Kerala,
India ^{1,2,3}

Assistant Professor, Dept. of Mechanical Engineering, Toc H Institute of Science and Technology, Ernakulam, Kerala,
India ⁴

ABSTRACT: Heat exchanger is an equipment for heat transfer from one medium to other. This paper deals with optimizations of shell and tube heat exchanger for maximum heat transfer, by the optimization of baffle cut & baffle angle of a particular shell and tube heat exchanger using CFD analysis. The performance of the shell and tube heat exchanger was studied by varying the parameters using CFD software package fluent. To validate the CFD algorithm the experiment was conducted on an existing single pass counter flow shell and tube heat exchanger. The optimum values obtained are baffle angle 5°, baffle cut 25%.

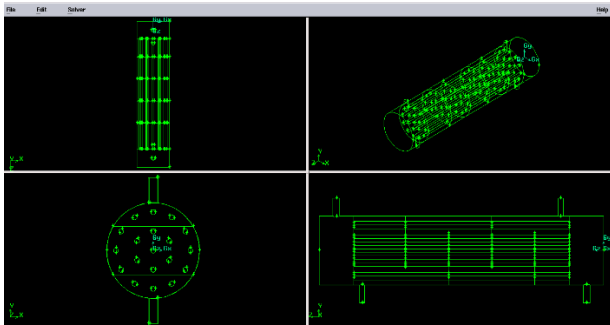
KEYWORDS: Shell and tube heat exchanger; Optimizations, CFD

I. INTRODUCTION

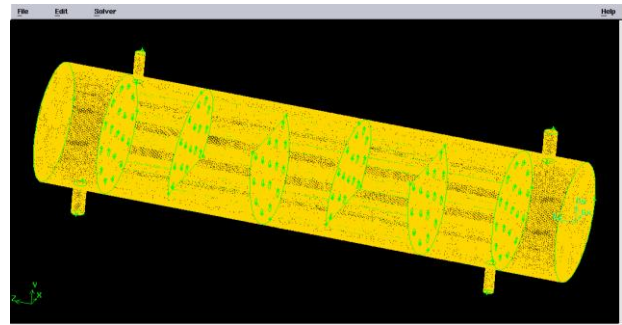
Heat exchangers are devices that facilitate the exchange of heat between two fluids that are at different temperatures while keeping them from mixing with each other. This device used to transfer heat between two or more fluids that are at different temperatures and which in most of the cases they are separated by a solid wall. Heat exchangers are used in power plants, nuclear reactors, refrigeration and air conditioning systems, automotive industries, heat recovery systems, chemical processing, and food industries. Different heat exchangers are named according to their applications. For example, heat exchangers being used to condense are known as condensers, similarly heat exchangers for boiling purposes are called boilers [1]. Shell and tube heat exchangers (STHXs) have been most widely used equipment in the industrial fields including: power plant, petroleum refining, steam generation etc. STHXs provide relatively large ratios of heat transfer area to volume and weight and can be easily cleaned [2]. Performance and efficiency of heat exchangers are measured through the amount of heat transferred using least area of heat transfer and pressure drop. Baffle Angle, Baffle Cut, Baffle Spacing, Tube Diameter and Number of Tubes These are the main factors affecting heat transfer of Heat exchanger. The optimization of shell-and-tube heat exchangers requires a good knowledge of the local and average shell-side heat transfer coefficients which is complicated by a shell diameter, baffle cut, baffle spacing, tube diameter, pitch, arrangement. These leakages reduce the velocity in the tube bundle and, hence, the heat transfer coefficient and pressure drop [3].

II. CFD ANALYSIS

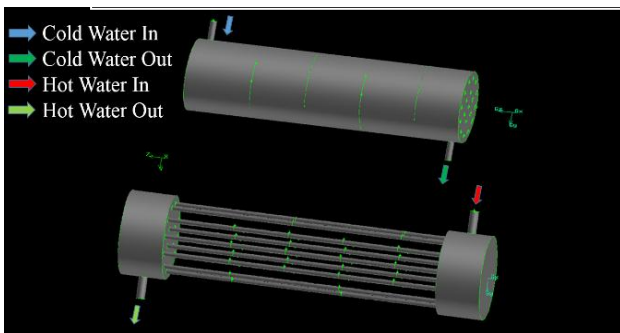
The fluid flow through the shell and heat transfer can be analysed using by CFD analysis. CFD (Computational Fluid Dynamics) it is a branch of that uses numerical methods and algorithms to solve and analyse problems that involve fluid flows. Computers are used to perform the calculations required to simulate the interaction of liquids and gases with surfaces defined by boundary conditions. CFD tools are used to predict the performance at various conditions. The flow characteristics such as pressure, volumetric flow, velocity and temperature at each point can be obtained from CFD tools. ANSYS FLUENT software is a high-performance, general purpose fluid dynamics program that has been applied to solve wide-ranging fluid flow problems. At the heart of ANSYS FLUENT is its advanced solver technology, the key to achieving reliable and accurate solutions quickly and robustly.



(a)



(b)



(c)

Boundary name	Type
Shell inlet	Velocity inlet
Shell outlet	Pressure outlet
Tube inlet	Velocity inlet
Tube outlet	Pressure outlet
Shell wall tube wall	Wall
Domain inside shell	Fluid
Domain inside tube	Fluid
Domain of baffles	Solid

(d)

Fig 1(a) shows the shell and tube heat exchanger before meshing, 1(b) shows the shell and tube heat exchanger after meshing, 1(c) shows the shell and tube heat exchanger boundaries and flow directions, 1(d) shows the boundary types and the continuum types given in the boundary conditions.

III. GOVERNING EQUATIONS

The Navier-Stokes equations are the basic governing equations for a fluid flow. It is obtained by applying Newton's second Law of Motion to a fluid element and is also called the momentum equation. It is supplemented by the mass conservation equation, also called continuity equation and the energy equation.

(i) Continuity Equation

In fluid dynamics, the continuity equation is an expression of conservation of mass. For a steady or unsteady incompressible 3 dimensional flow, the continuity equation can be expressed in Cartesian co-ordinates as:

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0 \quad (1)$$

Where u, v and w are the components of velocity field in x, y and z directions respectively.

(ii) Momentum Equation (Navier-Stokes Equation)

The Navier-Stokes equations are the basic governing equations for a fluid flow. It is obtained by applying Newton's Second Law of Motion to a fluid element and is also called the momentum equation. The Navier Stokes equation in scalar form for x, y and z directions are as follows.

a) Along x-direction

$$\left[\frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} + w \frac{\partial u}{\partial z} \right] \times \rho = -\frac{\partial p}{\partial x} + \rho g_x + \mu \left[\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2} \right] \quad (2)$$

b) Along y-direction

$$\left[\frac{\partial v}{\partial t} + u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} + w \frac{\partial v}{\partial z} \right] \times \rho = -\frac{\partial p}{\partial y} + \rho g_y + \mu \left[\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2} \right] \quad (3)$$

c) Along z-direction

$$\left[\frac{\partial w}{\partial t} + u \frac{\partial w}{\partial x} + v \frac{\partial w}{\partial y} + w \frac{\partial w}{\partial z} \right] \times \rho = -\frac{\partial p}{\partial z} + \rho g_z + \mu \left[\frac{\partial^2 w}{\partial x^2} + \frac{\partial^2 w}{\partial y^2} + \frac{\partial^2 w}{\partial z^2} \right] \quad (4)$$

Where p represents pressure, ρ is the density, μ is the dynamic viscosity and g is the acceleration due to gravity.

(iii) Energy Equation

$$\rho C_p = \left[\frac{\partial T}{\partial t} + u \frac{\partial T}{\partial x} + v \frac{\partial T}{\partial y} + w \frac{\partial T}{\partial z} \right] = k \left[\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial z^2} \right] + s \quad (5)$$

Where 'S' represent dissipation rate and 'k' is the thermal conductivity.

IV. CODE VALIDATION IN CFD & NUMERICAL METHODOLOGY

The experimental results were tabulated and the net heat transfer is calculated. And this result is later used for validating the algorithm used in CFD analysis. The algorithm used in this endeavour SIMPLE. It was validated by running the case with SST k-ω. k-ω gives better results when the flow is near to the wall. But when the flow reaches near to the bulk the value of the k-ω code has greater error. So SST model is used. SST model is a combination of k-ω and k-ε. k-ε code gives a better value in the bulk of the flow. So SST model is opted and then the value of the CFD results begins to be comparable to the results obtained from the experiments [4].

Inlet Conditions	
Cold inlet : Velocity inlet	= .708 m/s
Cold inlet : Temperature	= 303 K
Hot inlet : Velocity outlet	= 0.298 m/s
Hot inlet : Temperature	= 338 K
Outlet Conditions:	
Hot outlet : Pressure outlet	= P atm
Cold outlet : Pressure outlet	= P atm

(a)

Software package	FLUENT 6.3.26
Solver algorithm	SIMPLE (Semi Implicit Method for Pressure Linked Equation)
Turbulence model	K-omega SST
Grid sizes used for the grid independent study	1 mm 2 mm 3 mm 4 mm
optimum grid size	3.0mm

(b)

Fig 2(a) Shows the conditions given in the fluent for obtaining results. 2(b) shows the numerical methodology.

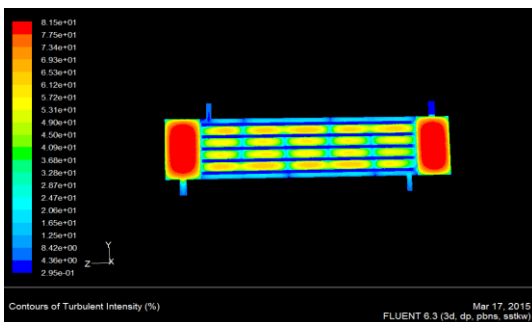
V. CFD RESULTS AND DISCUSSION

In CFD after code validation each parameters are changed and the net heat transfer for each specific design is calculated and is tabulated. From this result optimum value for each design changes is obtained and later this value is fixed and the next parameter is changed. At last an optimum design is calculated.

i) BAFFLE ANGLE

The initial baffle angle is 0° and later this is varied to 5°, 10°, 15°, and 20°. The maximum total heat transfer is 5047.68W and it is obtained for 5° baffle angle which is taken as optimum baffle angle.

Sl.no	Baffle Angle (degree)	Total heat transfer rate(W)	surface heat transfer coefficient (w/m2k)	Turbulent intensity (%)
1	0	4147.83	516.06	24.14%
2	5	5047.68	522.76	39.85%
3	10	3917.05	418.27	36.37%
4	15	4139.91	501.79	26.04%
5	20	3625.36	450.13	10.77%



(a)



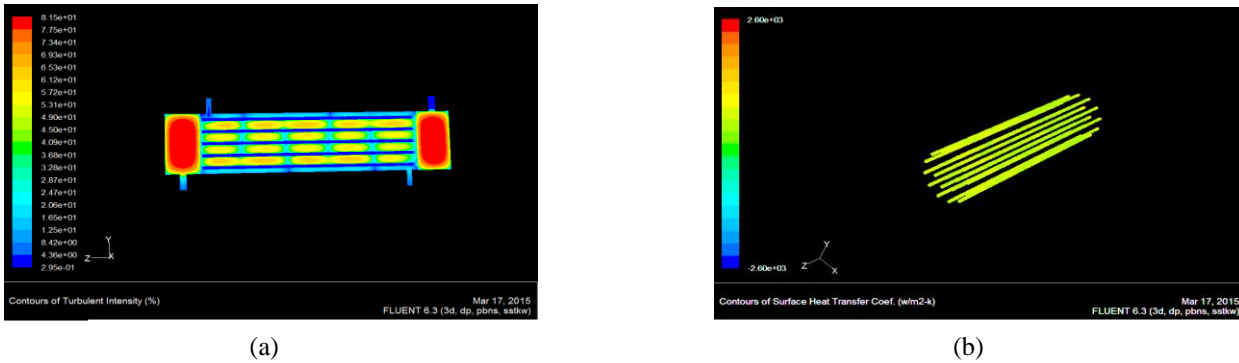
(b)

Fig 3(a) The turbulent intensity contour of the shell generated in fluent for 5° baffle angle, 3(b) The surface heat transfer coefficient contour of the tubes generated in the fluent for 5° baffle angle.

ii) BAFFLE CUT

The initial baffle cut is 20% and later this is varied to 20%, 25% and 30%. The maximum heat transfer is 5047.68W and it is obtained for 25% baffle cut and it is taken as optimum baffle cut.

Sl.no	Baffle Cut (%)	Total heat transfer rate(W)	surface heat transfer coefficient (w/m2k)	Turbulent intensity (%)
1	20	3952.38	508.05	17.28%
2	25	5047.68	522.76	39.85%
3	30	4214.23	479.17	26.38%



(a)

(b)

Fig 4(a) The turbulent intensity contour of the shell generated in fluent for 25% baffle angle, 4(b) The surface heat transfer coefficient contour of the tubes generated in the fluent for 25% baffle angle.

VI. CONCLUSION

Studied and analysed the fluid flow in a shell and tube heat exchanger and it is found to be complex, three dimensional (3D) and turbulent. The modelling and analysis was done in GAMBIT and FLUENT respectively. Optimization of shell and tube heat exchanger model for maximum heat exchanger has done using CFD analysis. After the optimization procedure, got 5° baffle angle and 25% Baffle cut be the optimized parameter. For both parameters, there is relation between heat transfer and turbulent intensity. Optimized model have maximum turbulent intensity than other arrangements.

REFERENCES

1. Usman Ur Rehman, Heat Transfer Optimization of Shell-and-Tube Heat Exchanger through CFD Studies, Master's Thesis in Innovative and Sustainable Chemical Engineering, Göteborg, Sweden, 13-14, 2011
2. Vijaykumar Chalwa, Nishal Kadli, Study Of Variation For Pressure Drop And Temperature Distribution In A Shell And Tube Heat Exchanger In Case Of Vertical Baffle, *Mechanica Confab*, 16-19, 2013
3. Huadong Li And Volker Kottke, Effect Of The Leakage On Pressure Drop And Local Heat Transfer In Shell-And-Tube Heat Exchangers For Staggered Tube Arrangement, *Int. J. Heat Mass Transfer*, Elsevier Science Ltd., 325-329, 1998
4. KEVIN M. LUNSFORD, Increasing Heat Exchanger Performance, *Bryan Research and Engineering Technical Papers*, 1998
5. Simin Wang, Jian Wen, Yanzhong Li, an experimental investigation of heat transfer enhancement for a shell and tube heat exchanger, *Applied thermal engineering*, 2008
6. Yan Li, Xiumin Jiang, Xiang yong Huang, Jigang Jia, Jianhui Tong, Optimization of high pressure shell and tube heat exchanger for syngas cooling in a IGCC, *IJHM*, 2010
7. Farhad nemati taheer, kazem razmi et al, Baffle space impact on the performance of helical baffle shell and tube heat exchangers, *applied thermal engineering*, 2011
8. Xiaoming Xiao, youmei xia et al, Numerical investigation of helical baffles heat exchanger with different Prandtl number fluids, *IJHM*, 2012
9. Yonghua you, Wei liu et al, Numerical modelling and experimental validation of heat transfer and flow resistance on the shell side of a shell-and-tube heat exchanger with flower baffles, *IJHM*, 2012
10. M.M. Elias, I.M. Shahrul, I.M. Mahbul, R. Saidur, N.A. Rahim, Effect of different nano-particle shapes on shell and tube heat exchanger using different baffle angles and operated with nanofluid, *IJHM*, 2013
11. Rajiv Mukherjee, Effectively Design Of Shell-and-Tube Heat Exchangers.