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

Joemer.C.S\textsuperscript{1}, Sijo Thomas\textsuperscript{2}, Rakesh.D\textsuperscript{3}, Nidheesh.P\textsuperscript{4}

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

Assistant Professor, Dept. of Mechanical Engineering, Toc H Institute of Science and Technology, Ernakulam, Kerala, India\textsuperscript{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.
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:

\[
\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0
\]  

(1)

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

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

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

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

Where \( p \) represents pressure, \( \rho \) is the density, \( \mu \) is the dynamic viscosity and \( g \) is the acceleration due to gravity.

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

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-\( \omega \). k-\( \omega \) 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-\( \omega \) code has greater error. So SST model is used. SST model is a combination of k-\( \omega \) and k-\( \epsilon \). k-\( \epsilon \) 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].

<table>
<thead>
<tr>
<th>Inlet Conditions</th>
<th>Software package</th>
<th>FLUENT 6.3.26</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cold inlet : Velocity inlet = .708 m/s</td>
<td>Solver algorithm</td>
<td>SIMPLE (Semi Implicit Method for Pressure Linked Equation)</td>
</tr>
<tr>
<td>Cold inlet : Temperature = 303 K</td>
<td>Turbulence model</td>
<td>K-omega SST</td>
</tr>
<tr>
<td>Hot inlet : Velocity outlet = 0.298 m/s</td>
<td>Grid sizes used for the grid independent study</td>
<td>1 mm</td>
</tr>
<tr>
<td>Hot inlet : Temperature = 338 K</td>
<td>optimum grid size</td>
<td>3.0mm</td>
</tr>
<tr>
<td>Outlet Conditions:</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Hot outlet : Pressure outlet = P atm</td>
<td>2 mm</td>
<td></td>
</tr>
<tr>
<td>Cold outlet : Pressure outlet = P atm</td>
<td>3 mm</td>
<td></td>
</tr>
<tr>
<td></td>
<td>4 mm</td>
<td></td>
</tr>
</tbody>
</table>

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

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

![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.](image)

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.

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

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 taher, kazem razni 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