Design and CFD Analysis of Shell and Tube Heat Exchanger
Mohammed Irshad1, Mohammed Kaushar2, G. Rajmohan3
Student1,2, Assistant Professor3
Department of Mechanical Engineering
Dhanish Ahmed College of Engineering Chennai, India

Abstract:
Shell and tube heat exchangers are the most common type of heat exchangers used in present scenario. Heat exchangers are widely used equipment in various industries such as power generation and transportation, refrigeration industry and chemical process industries because it suits high pressure application. Presented in this project is comparison for several shell- and tube heat exchangers with segmental baffles. The objective of this project is to design a shell and tube heat exchanger with segmental baffles and to study the flow and temperatures inside the shell and tubes using ANSYS software tool for the different baffles assemblies and orientation also overall heat transfer is calculated for each design. This project totally contains 5 designs for comparison. The process in solving simulation consists of modelling and meshing the basic geometry of shell and tube heat exchanger using CFD package ANSYS 14.5. The heat exchanger contains 7 tubes and 600mm length and 90mm shell diameter. In simulation we will show how the temperature, pressure, velocity varies in shell due to different baffles orientation.

Keywords: Shell and tube heat exchanger, Solid works, Ansys CFD Fluent, Segmental baffles, outlet pressure difference.

I. INTRODUCTION
A ‘heat exchanger’ may be defined as an equipment which transfers the energy from a hot fluid to a cold fluid, with maximum rate and minimum investment and running costs. In heat exchanger the temperature of each fluid changes as it passes through the exchangers, and hence the temperature of the dividing wall between the fluids also changes along the length of the exchanger. One of the important processes in engineering is the heat exchanger between flowing fluids, and many types of heat exchangers are employed in various types of installations, as petro-chemical plants, process industries, pressurised water reactor power plants, nuclear power stations and refrigeration systems. On the basis of design tubular or shell and tube type of heat exchangers are widely in use. The shell and tube heat exchangers are the one in which one of the fluids flows through a bundle of tubes enclosed by a shell. The other fluid is forced through the shell and it flows over the outside surface of the tubes. Such an arrangement is employed where reliability and heat transfer effectiveness are important. With the use of multiple tubes heat transfer rate is amply improved due to increased surface area. These heat exchanger’s larger heat transfer surface area-to-volume ratios than the most of common types of heat exchangers, and they are manufactured easily for a large variety of sizes and flow configurations. They can operate at high pressures, and their construction facilities disassembly for periodic maintenance and cleaning.

II. LITERATURE SURVEY
There were many works previously carried out on the Shell and tube heat exchangers. Some of them are enlisted here:

A. Experimental performance comparison of shell side heat transfer for shell-and-tube heat exchangers with middle-overlapped helical baffles & segmental baffles-Jian-Fei Zhang et-al
Presented in this paper are experimental test and comparison for several shell-and-tube heat exchangers, one with segmental baffles and four with helical baffles at helix angles of 20°, 30°, 40° and 50°, respectively. The orientation of the segmental baffles are kept constant that is 90° vertical.[1]

B. Experimental investigation of shell-and-tube heat exchanger with different type of baffles- Pooja J. Pawar et al
This paper focus on the experimental investigation of shell and tube heat exchanger with different type of baffles. The shell and tube heat exchanger with segmental baffles and flower baffles are designed, fabricated and tested. The heat exchanger with flower baffle gives more efficient overall performance up to 25-32% than segmental baffles heat exchanger. Also the pressure drop gets reduced in flower baffle heat exchanger up to 20-28% than segmental baffles heat exchanger. [2]

C. Design and Performance Study of Shell and Tube Heat Exchanger with Single Segmental Baffle Having Perpendicular & Parallel-Cut Orientation-Swarup S Deshpande et al
This paper primarily focuses on the design and comparative analysis of Single segmental Shell and tube Heat Exchanger with perpendicular & parallel baffle cut orientation. For designing Kern Method is used. It predicts heat transfer coefficient, Pressure drop of both arrangements. This method gives us clear idea that rate of heat transfer is greater in Perpendicular-cut baffle orientation than Parallel-cut, Pressure drop approximately remaining same. [3]

D. Shell side CFD analysis of a small shell-and-tube heat exchanger - Ender Ozden et al
The author portrays that the heat transfer coefficient and the pressure drop are dependent on the baffle spacing, baffle cut and shell diameter. This is investigated by numerically modelling a small heat exchanger. The flow and temperature fields inside the shell are resolved using a commercial CFD package. A set of CFD simulations is performed for a single shell and single tube pass heat exchanger with a variable number of baffles and turbulent flow. The results are observed to be sensitive to the turbulence model selection. The best turbulence model among the ones considered is determined by comparing the CFD
results of heat transfer coefficient, outlet temperature and pressure drop with the Bell–Delaware method results. For two baffle cut values, the effect of the baffle spacing to shell diameter ratio on the heat exchanger performance is investigated by varying flow rate. [4]

E. An experimental investigation of heat transfer enhancement for a shell-and-tube heat exchanger- Simin Wang et al

For the purpose of heat transfer enhancement, the configuration of a shell-and-tube heat exchanger was improved through the installation of sealers in the shell-side. The gaps between the baffle plates and shell is blocked by the sealers, which effectively decreases the short-circuit flow in the shell-side. The results of heat transfer experiments show that the shell-side heat transfer coefficient of the improved heat exchanger increased by 18.2–25.5% .the overall coefficient of heat transfer increased by 15.6–19.7% and Pressure losses increased by 44.6–48.8% with the sealer installation, but the increment of required pump power can be neglected compared with the increment of heat flux. [5]

III. INTRODUCTION TO CAD/CAM/CAE

The possible basic way to industries is to have high quality products at low cost is by using the computer Aided Engineering (CAE), Computer Aided Design (CAD) And Computer Aided Manufacturing (CAM) set up. Further many tools is been introduced to simplify & serve the requirement Catia, Pro-E, UG are among many.

IV. MODELLING, MESHING AND ANALYSIS OF SHELL AND TUBE HEAT EXCHANGER

In this study shell and tube heat exchanger with six baffles are placed along the shell in alternating orientations with cut facing up, cut facing down, etc., in order to create flow paths across tube bundle. The geometric model is compared by varying baffle inclination i.e 0°,20°, 30°, and 40°. The CFD analysis involves pre-processing, solving and post processing. The geometry modelling is done using software called Solid Works 15.0 as it is easy to model Heat exchanger in 3D modelling software.

Table 1. Geometric dimnesion of Heat exchanger

<table>
<thead>
<tr>
<th>SPECIFICATION</th>
<th>DIMENSIONS</th>
</tr>
</thead>
<tbody>
<tr>
<td>Length of heat exchanger, L</td>
<td>600 mm</td>
</tr>
<tr>
<td>Shell inner diameter, D_s</td>
<td>90 mm</td>
</tr>
<tr>
<td>Tube length, l</td>
<td>600 mm</td>
</tr>
<tr>
<td>Tube outer diameter, d_o</td>
<td>20 mm</td>
</tr>
<tr>
<td>No. of tubes, N_t</td>
<td>07</td>
</tr>
<tr>
<td>Tube pitch &amp; geometry, P_t</td>
<td>30, triangular</td>
</tr>
<tr>
<td>Baffle inclination, ( \theta )</td>
<td>0°,20°,30°,40°</td>
</tr>
<tr>
<td>Baffle cut</td>
<td>36%</td>
</tr>
<tr>
<td>Baffle spacing, ( \Delta B_b )</td>
<td>86 mm</td>
</tr>
<tr>
<td>Baffles thickness, t</td>
<td>03 mm</td>
</tr>
<tr>
<td>No. of baffles N_b</td>
<td>6</td>
</tr>
</tbody>
</table>

GOVERNING EQUATIONS

The 3-D flow through the shell-and-tube heat exchanger has been simulated by solving the appropriate governing equations, eq. (1) to eq. (5), viz. conservation of mass, momentum and energy using ANSYS CFX code. Turbulence is taken care by shear stress transport (SST) k-w model of closure which has a blending function that supports Standard k-w near the wall and Standard k-e elsewhere.

GEOMETRY MODELING

First the geometry of the model is created in SOLIDWORKS 15. The model is saved in Para solid type i.e. (.xt). The external geometry file is imported in the design modeller of the ansys fluent. The geometry has totally 08 parts. One shell and 7 tubes bundle.

Figure 1. Geometry of the model

MESHING

In free meshing a relatively coarse mesh is generated. It contains both tetrahedral and hexahedral cells having triangular and quadrilateral faces at the boundaries. Later, a fine mesh is generated using edge sizing. In this, the edges and regions of high pressure and temperature gradients are finely meshed.

Figure 2. Mesh of the model

BOUNDARY CONDITIONS:

Different boundary conditions were applied for different zones. Since it is a shell-and-tube heat exchanger, there are two inlets and two outlets. The inlets were defined as velocity inlets and outlets were defined as pressure outlets. The inlet velocity of the cold fluid was kept constant i.e. 0.0787m/s, whereas velocity of hot fluid was kept constant i.e. 1.594 m/s. The outlet pressures were kept default i.e. atmospheric pressure. The hot fluid temperature at inlet was 340k and cold fluid inlet
temperature was kept 300k. The other wall conditions were defined accordingly. The surrounding air temperature was kept 300k.

**SOLUTION METHODS:**
The solution methods were set as follows:
- Scheme = Simple
- Gradient = Least Square Cell Based
- Pressure = linear
- Momentum = Second Order Upwind
- Turbulent Kinetic Energy = Second Order Upwind
- Turbulent Dissipation Rate = Second Order Upwind
- Energy = power law

**SOLUTION CONTROL:**
Under relaxation factors the parameters are:
- Pressure = 0.7 Pa
- Density = 1 kg/m³
- Body forces = 1 kg/m³ s²
- Momentum = 0.2 kg m/s
- Turbulent kinetic energy = 1 m²/s²

**SOLUTION INITIALIZATION:**
Pressure Velocity coupling selected as SIMPLEC. Skewness correction was set at 0. In Spatial Discretization zone Gradient was set as “Least square cell based”. Pressure was “standard”.
- Momentum was “First order Upwind”.
- Turbulent Kinetic energy was set as “First order Upwind”.
- Energy was also set as “First order Upwind”.

Solution initialization was “standard method” and solution was initialize from inlet with 300k temperature. Under the Above boundary condition and solution initialize condition simulation was set for 100 iteration.

**V. RESULTS AND DISCUSSION**
The computational fluid dynamics analysis was done for the five types of STHXs, in which cold water flow through the shell side and hot water flow through the tube side of the STHXs. Performance of two STHXs are compared.

**Validation:**
Simulation results are obtained for 0.5 kg/s fluid flow rate for the model with 0° baffle inclination angle are validated with the data available in [4]. It is found that the exit temperature at the shell outlet is matching with the literature results and the deviation between the two is 0.04% to 1%. The simulation results for 0.5 kg/s mass flow rate for models with 0°, 20°, 30°, and 40° baffle inclination are obtained. It is seen that the temperature gradually increase from 300 k at the inlet to 340K at the outlet of the shell side. The average temperature at the outlet surface is nearly 330K. There is no much variation of temperature for all five cases considered. The maximum pressure for models with 0°, 20°, 30°, and 40° baffle inclination are 17.84, 11.29, 19.03, 20.83, 22.69 Pa, respectively. The maximum velocity is nearly equal to 0.11 m/s for all five models at the inlet and exit surface and the velocity magnitude reduces to zero at the baffles surface. It can be seen that compared to 0°, 20°, 40° baffle inclination angle, 30° baffle inclination angle provides a smoother flow with higher heat transfer coefficient. Also it is found that the within the operating conditions, the shell- side heat transfer coefficient of the without short –circuit flow heat exchanger is 19.61% more than the heat exchanger with short –circuit flow between shell and baffle gap. This is validated by the data available in literature [5].

**TEMPERATURE VARIATION**

![Figure 4. 0° baffle orientation without leakage](image)

![Figure 5. 0° baffle orientation with leakage](image)
**HEAT TRANSFER RATE**

\[
Q = \dot{m}_h \cdot C_p_h \cdot \Delta T_h = \dot{m}_c \cdot C_p_c \cdot \Delta T_c
\]

- \( m \) = mass flow rate
- \( C_{p_h,c} \) = specific heat of fluids
- \( \Delta T_{h,c} \) = Temperature difference between shell and tube sides

**Table 3. Total Heat transfer rate**

<table>
<thead>
<tr>
<th>Baffle inclination angle(degree)</th>
<th>Total Heat Transfer rate Q (W)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0° inclination</td>
<td>66,848</td>
</tr>
<tr>
<td>0° with shell and baffle gap</td>
<td>60,581</td>
</tr>
<tr>
<td>20° inclination</td>
<td>70,608</td>
</tr>
<tr>
<td>30° inclination</td>
<td>74,368</td>
</tr>
<tr>
<td>40° inclination</td>
<td>77,502</td>
</tr>
</tbody>
</table>

**LOGARITHMIC MEAN TEMPERATURE DIFFERENCE**

\[
(\Delta T)_m = \frac{[T_1 - t_1] - (T_2 - t_2)}{\ln(T_1/t_1)}
\]

**Table 4. Logarithmic Mean Temperature Difference**

<table>
<thead>
<tr>
<th>Baffle inclination angle(degree)</th>
<th>Logarithmic Mean Temperature Difference (LMTD) (( \Delta T ))_m°C</th>
</tr>
</thead>
<tbody>
<tr>
<td>0° inclination</td>
<td>19.31</td>
</tr>
<tr>
<td>0° with shell and baffle gap</td>
<td>21.72</td>
</tr>
<tr>
<td>20° inclination</td>
<td>17.27</td>
</tr>
<tr>
<td>30° inclination</td>
<td>15.24</td>
</tr>
<tr>
<td>40° inclination</td>
<td>10.02</td>
</tr>
</tbody>
</table>

**OVERALL HEAT TRANSFER CO-EFFICIENT (U) W/m²K**

\[
Q = U \cdot A \cdot (\Delta T)_m
\]

- \( U \) – Overall heat transfer co-efficient W/m²K
- \( A \) – cross-sectional area m²
- \( (\Delta T)_m \) – logarithmic mean temperature difference

**Table 5. Overall heat Transfer Coefficient**

<table>
<thead>
<tr>
<th>Baffle inclination angle(degree)</th>
<th>CFD Overall heat transfer co-efficient U W/m²K</th>
</tr>
</thead>
<tbody>
<tr>
<td>0° inclination</td>
<td>2498</td>
</tr>
<tr>
<td>0° with shell and baffle gap</td>
<td>2026</td>
</tr>
<tr>
<td>20° inclination</td>
<td>3397</td>
</tr>
<tr>
<td>30° inclination</td>
<td>6088</td>
</tr>
<tr>
<td>40° inclination</td>
<td>6123</td>
</tr>
</tbody>
</table>

**VELOCITY VARIATION**

As the mass flow rate for all the assemblies i.e. 0°, 20°, 30°, 40° is 0.5kg/s therefore much variation in velocity is not noticed. The maximum velocity at inlet is noticed as an average of 0.11086 m/s and the minimum is 0.0m/s at exit.

**PRESSURE VARIATION:**

<table>
<thead>
<tr>
<th>Baffle inclination angle(degree)</th>
<th>Max pressure Pa</th>
<th>Min pressure Pa</th>
<th>Pressure Difference Pa</th>
</tr>
</thead>
<tbody>
<tr>
<td>0° without leakage</td>
<td>20.83</td>
<td>-2.313</td>
<td>23.143</td>
</tr>
<tr>
<td>0° with leakage</td>
<td>11.29</td>
<td>-2.28</td>
<td>13.57</td>
</tr>
<tr>
<td>20° inclination</td>
<td>19.03</td>
<td>-2.558</td>
<td>21.588</td>
</tr>
<tr>
<td>30° inclination</td>
<td>17.84</td>
<td>-2.63</td>
<td>20.47</td>
</tr>
<tr>
<td>40° inclination</td>
<td>16.96</td>
<td>-1.653</td>
<td>18.613</td>
</tr>
</tbody>
</table>

**V. CONCLUSION**

The heat transfer and flow distribution is discussed in detail. From CFD simulation results, for fixed tube wall and shell inlet temperatures, shell side heat transfer coefficient, pressure...
drop and heat transfer rate values are obtained. From the CFD result it is observed that the heat exchanger without any short-circuited flow has the higher heat transfer coefficient than the heat exchanger with leakage. It’s found that the overall heat transfer coefficient increases by 18.89% if the sealers are installed inside the shell and tube heat exchanger. It is found that for 0.5 kg/s mass flow rate there is no much effect on outlet temperature of the tube even though the baffle inclination is increased from 0° to 40°. However the shell-side pressure difference is decreased with increase in baffle inclination angle i.e., as the inclination angle is increased from 0° to 40°. The pressure difference is decreased by 6%, for the heat exchanger with 20° baffle inclination angle and by 19.57% for the heat exchanger with 40° baffle inclination angle with 36% baffle cut. Baffle cut is reduced in order to provide proper support to the centre row of tubes. It is noticed that for the 36% baffle cut only 20° baffle inclination angle is maximum. If the angle is beyond 20°, the centre row of tubes is not supported. Hence the baffle cut can’t be used effectively. Also for the given geometry the mass flow rate must be below 2 kg/s, if it is increased beyond 2 kg/s the pressure drop increases rapidly with little variation in outlet temperature. Hence it can be concluded that shell and tube heat exchanger with 40° inclination angle and 25% baffle cut results in better performance compared to 0°, 20° and 30° inclination angle.

VI. FUTURE SCOPE

In this project, a solution method of the shell and tube heat exchanger design optimization problem was proposed based on the utilization of CFD. Referring to the literature test cases, reduction of capital investment up to 7.4% and savings in operating costs up to 93% were obtained, with an overall decrease of total cost up to 52%, showing the improvement potential of the proposed method. Furthermore, the genetic algorithm allows for rapid solution of the design problem and enables to examine. A number of alternative solutions of good quality, giving the designer more degrees of freedom in the final choice with respect to traditional methods. As a future work, it intended to deal in detail with issues of mechanical design and Creation of CAD model is done by using CAD modeling software (CATIA V5). Design of the shell and tube heat exchanger with required parameters like external diameter, internal diameter, and baffle space is shown through CATIA V5. By studying the properties of the varied materials, I manage to change the properties of the materials in the shell and tube to increase the heat transfer rate and cost efficient process through CFD Analysis.

VII. REFERENCES


