SHELL SIDE CFD ANALYSIS OF A SMALL SHELL-AND-TUBE HEAT EXCHANGER CONSIDERING THE EFFECTS OF BAFFLE INCLINATION ON FLUID FLOW

Abdur Rahim 1*, S.M.Saad Jameel 2
Department of Mechanical Engg, Faculty of Engineering and Technology, Jamia Millia Islamia, New Delhi – 110025, India.
E-mail: 1rahim_ark@rediffmail.com, 2saadj11@gmail.com

Abstract
The shell side design of a shell-and-tube heat exchanger; in particular the baffle cut and baffles inclination dependencies of the heat transfer coefficient and the pressure drop are investigated by numerically modeling a small heat exchanger. The flow and temperature fields inside the shell are resolved using a commercial CFD software tool STAR CCM+ v6.06. In this present work, attempts were made to investigate the impacts of various baffle inclination angles on fluid flow and the heat transfer characteristics of a shell-and-tube heat exchanger for three different baffles inclination angles namely 0°, 10° and 20°. The simulation results for various shell and tube heat exchangers, one with segmental baffles perpendicular to fluid flow and two with segmental baffles inclined to the direction of fluid flow are compared for their performance. The results are observed to be sensitive to the turbulence model selection. For a given baffle cut of 36 %, the heat exchanger performance is investigated by varying mass flow rate and baffle inclination angle. From the CFD simulation results, the shell side outlet temperature, pressure drop, recirculation near the baffles, heat transfer, optimal mass flow rate and the optimum baffle inclination angle for the given heat exchanger geometry are determined.

Keywords: Shell-and-tube heat exchanger, CFD, Conjugate Heat Transfer, Pressure drop, Baffle inclination angle, turbulence models.

1. Introduction
Heat exchangers have always been an important part to the lifecycle and operation of many systems. A heat exchanger is a device built for efficient heat transfer from one medium to another in order to carry and process energy. Typically one medium is cooled while the other is heated. They are widely used in petroleum refineries, chemical plants, petrochemical plants, natural gas processing, Air conditioning, refrigeration and automotive applications. One common example of a heat exchanger is the radiator in a car, in which it transfers heat from the water (hot engine-cooling fluid) in the radiator to the air passing through the radiator. There are two main types of heat exchangers.

• Direct contact heat exchanger, where both media between which heat is exchanged are in direct contact with each other.
• Indirect contact heat exchanger, where both media are separated by a wall through which heat is transferred so that they never mix.

Shell and tube type heat exchanger is an indirect contact type heat exchanger as it consists of a series of tubes, through which one of the fluids runs. The shell is a container for the shell fluid. Usually, it is cylindrical in shape with a circular cross section, although shells of different shapes are used in specific applications. For this particular study E shell is considered, which is generally a one pass shell. E shell is the most commonly used due to its low cost and simplicity, and has the highest log-mean temperature-difference (LMTD) correction factor. Although the tubes may have single or multiple passes, there is one pass on the shell side, while the other fluid flows within the shell over the tubes to be heated or cooled. Shell-and-tube heat exchangers in various sizes are widely used in industrial operations and energy conversion systems. Tubular Exchanger Manufacturers Association (TEMA) regularly publishes standards and design recommendations.

The tube side and shell side fluids are separated by a tube sheet, Gaddis [1], Schlunder [2], Mukherjee [3]. The heat exchanger model used in this study is a small sized one, as compared to the main stream, all of the leakage and bypass streams do not exist or are negligible, Ender Ozden and Ilker Tari [4] , Uday Kapale and Satish Chand [5], Thirumarimurugan et al. [6]. Baffles are used to support the tubes for structural rigidity, preventing tube vibration and sagging and to divert the flow across the bundle to obtain a higher heat transfer coefficient. Baffle spacing (B) is the centre line distance between two adjacent baffles, Sparrow and Reischneider [7], Li and Kottke [8], Su Thet Mon Than et al. [9]. Baffle is provided with a cut (Bc) which is expressed as the percentage of the segment height to shell inside diameter. Baffle cut can vary between 15% and 45% of the shell inside.
diameter, Kakac and Liu [10], Gay et al. [11], Emerson [12]. In the present study 36% baffle cut (Bc) is considered. In general, conventional shell and tube heat exchangers result in high shell-side pressure drop and formation of recirculation zones near the baffles. Most of the researches now a day are carried on helical baffles, which give better performance then single segmental baffles but they involve high manufacturing cost, installation cost and maintenance cost. The effectiveness and cost are two important parameters in heat exchanger design. So, in order to improve the thermal performance at a reasonable cost of the Shell and tube heat exchanger, baffles in the present study are provided with some inclination in order to maintain a reasonable pressure drop across the exchanger Yong-Gang Lei et al. [13]. The complexity with experimental techniques involves quantitative description of flow phenomena using measurements dealing with one quantity at a time for a limited range of problem and operating conditions. Computational Fluid Dynamics is now an established industrial design tool, offering obvious advantages Versteeg and Malalasekera [14].

In this study, a full 360° CFD model of shell and tube heat exchanger is considered. By modeling the geometry as accurately as possible, the flow structure and the temperature distribution inside the shell are obtained. In this study, a small shell-and-tube heat exchanger is modeled for CFD simulations. A commercial CFD package, STAR CCM+ version6 [16], is used together with Hyper Mesh for mesh generation software. Sensitivity of the simulation results to modeling choices such as mesh and turbulence model is investigated. After selecting a suitable mesh, a discretization scheme and a turbulence model, simulations are performed for two different shell side flow rates by varying baffle inclination and 36 % baffle cut. The simulation results are used for calculating shell side heat transfer coefficient and pressure drop.

2. Modeling Details
In this study, a small heat exchanger is selected in order to increase the model detail and to make solid observations about the flow inside the shell. Some of the design parameters and the predetermined geometric parameters are presented in Table 1. The geometric model with six baffles is shown in Figure 1 36% baffle cut value is selected to place the cut just below or above the central row of tubes. The working fluid of the shell side is water. Since the properties of water are defined as constants in the STARCCM+ database, to improve the accuracy; they are redefined using piecewise-linear functions of temperature by using the “Thermo-Physical Properties of Saturated Water” tables available in the literature [15].

In this study six baffles are placed along the shell in alternating orientations with cut facing up, cut facing down, cut facing up again etc, in order to create flow paths across the tube bundle. The geometric model is optimized by varying the baffle inclination angle i.e., 0°, 10° and 20°. The computational modeling involves pre-processing, solving and post-processing. The geometry modeling of shell and tube heat exchanger is explained below.

2.1. Geometry modeling
The model is designed according to TEMA (Tubular Exchanger Manufacturers Association) Standards Gaddis (2007), using STARCCM+ software as shown in Fig. 1. Design parameters and fixed geometric parameters have been taken similar to Ender Ozden and Ilker Tari (2010), as indicated in Tab. 1.

Figure 1. Isometric view of arrangement of baffles and tubes of shell and tube heat exchanger with 10° baffle inclination.

Table 1. Geometric dimensions of shell and tube heat exchanger.
### 2.2 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. The resulting equations are:

**Conservation of mass:** \( \rho \dot{V} = 0 \) \hspace{1cm} (1)

**x-momentum:** \( \nabla \cdot (\rho u V_r) = -\partial p/\partial x + \partial \tau_{xx}/\partial x + \partial \tau_{xy}/\partial y + \partial \tau_{xz}/\partial z \) \hspace{1cm} (2)

**y-momentum:** \( \nabla \cdot (\rho v V_r) = -\partial p/\partial y + \partial \tau_{xy}/\partial x + \partial \tau_{yy}/\partial y + \partial \tau_{yz}/\partial z + \rho g \) \hspace{1cm} (3)

**z-momentum:** \( \nabla \cdot (\rho w V_r) = -\partial p/\partial z + \partial \tau_{xz}/\partial x + \partial \tau_{yz}/\partial y + \partial \tau_{zz}/\partial z + \rho g \) \hspace{1cm} (4)

**Energy:** \( \nabla \cdot (\rho e V_r) = -p \nabla \cdot V_r + \nabla \cdot (k \nabla T) + q + \Phi \) \hspace{1cm} (5)

In Eq. (5), \( \Phi \) is the dissipation function that can be calculated from

\[
\Phi = \mu \left[ 2[(\partial u/\partial x)^4 + (\partial v/\partial y)^4 + (\partial w/\partial z)^4] + (\partial u/\partial y + \partial v/\partial x)^4 + (\partial u/\partial z + \partial w/\partial x)^4 + (\partial v/\partial z + \partial w/\partial y)^4 \right] + \lambda (\nabla V_r)^2 \hspace{1cm} (6)
\]

### 2.3 Boundary conditions

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

### 2.4 Mesh selection

Mesh generation is performed using STARCCM+. The surfaces of the model are meshed using triangular elements. The shell volume is meshed using tetragonal elements. Mesh size selected for six baffle case: the coarse mesh with approximately 292388 elements have taken. The entire geometry is divided into three fluid domains Fluid_Inlet, Fluid_Shell and Fluid_Outlet and six solid domains namely Solid_Baffle1 to Solid_Baffle6 for six baffles respectively.

### 2.5 Turbulence model

Since the flow in this study is turbulent, turbulence effects should be taken into account using turbulence modeling. The choice of turbulence model is very critical in CFD simulations. However, there is no universal criterion for selecting a turbulence model.
In this study, k-ε turbulence model are tried. The standard k-ε model is a semi-empirical model based on model transport equations for the turbulence kinetic energy k and its dissipation rate ε. For steady state, k and ε are obtained from the following transport equations:

$$\frac{\partial}{\partial x_i} (\rho k u_i) = \frac{\partial}{\partial x_j} \left[ (\mu + \mu_s) \frac{\partial k}{\partial x_j} \right] + G_k + G_b - \rho \varepsilon + S_k$$

(7)

$$\frac{\partial}{\partial x_i} (\rho \varepsilon u_i) = \frac{\partial}{\partial x_j} \left[ (\mu + \mu_s) \frac{\partial \varepsilon}{\partial x_j} \right] + C_{1\varepsilon} \frac{\varepsilon}{k} (G_k + C_3 G_b) - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k} + S_k$$

(8)

And the turbulent viscosity is defined by the following equation:

$$\mu_t = \rho C_{\mu} \frac{k^{2/3}}{\varepsilon}$$

(9)

The model constants have the following values:

$$C_{1\varepsilon} = 1.44, \ C_{2\varepsilon} = 1.92, \ C_{\mu} = 0.09, \ \sigma_k = 1, \ \sigma_\varepsilon = 1.3$$

### 3. Results and Discussion

#### 3.1 Validation

Simulation results are obtained for different mass flow rates of shell side fluid ranging from 1 kg/s and 2 kg/s. The simulated results for 1 kg/s fluid flow rate for model with 0° baffle inclination angle are validated with the data available in the literature Ender Ozden and Ilker Tari (2010). It is found that the exit temperature at the shell outlet is matching with the literature results and the deviation between the two is less than 1%.

The simulation results for 1 kg/s mass flow rate for models with 0°, 10° and 20° baffle inclination are obtained. It is seen that the temperature gradually increases from 300 K at the inlet to 330 K at the outlet of the shell side. The average temperature at the outlet surface is nearly 326 K for all the three models. There is no much variation of temperature for all the three cases considered. The maximum pressure for models with 0°, 10° and 20° baffle inclinations are 17220.5, 8474.66 and 13705.9 Pascal respectively. The pressure drop is less for 20° baffle inclination compared to other two models due to smoother guidance of the flow. The maximum velocity is nearly equal to 1.595 m/s for all the three 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° baffle inclination angle, 10° & 20° baffle inclination angles, provide a smoother flow with the inclined baffles guiding the fluid flow.

From the stream line contour of Fig. 3-5, it is found that recirculation near the baffles induces turbulence eddies which would result in more pressure drop for model with θ = 0° whereas as recirculation are lesser for model with θ = 10° and the recirculation formed for model with θ = 20° are much less in comparison to the other two models which indicates the resulting pressure drop is optimum as shown in Fig. 6.

Table 3. Outlet temperature, Shell-side pressure drop values, Heat transfer coefficient for various baffle inclinations and mass flow rates.

<table>
<thead>
<tr>
<th>Baffles Inclination Angle (degree)</th>
<th>Shell side Outlet temp (K)</th>
<th>Shell side Mass flow rate = 1 kg/s</th>
<th>Heat transfer coeff. (W/m²K)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0°</td>
<td>325.65</td>
<td>3210.46</td>
<td>106094.6</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>2928.4</td>
</tr>
<tr>
<td>10°</td>
<td>324.69</td>
<td>3082.04</td>
<td>104046.9</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>2872.79</td>
</tr>
<tr>
<td>20°</td>
<td>329.75</td>
<td>2356.2</td>
<td>123318</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>3474.12</td>
</tr>
</tbody>
</table>

From the CFD simulation results, for fixed tube wall and shell inlet temperatures, shell side outlet temperature, pressure drop values, heat transfer and heat transfer coefficient for varying fluid flow rates are provided in Table 3 and it is found that the shell outlet temperature decreases with increasing mass flow rates as expected even the variation is minimal. It is found that for three mass flow rates1 kg/s & 2 kg/s there is no much effect on outlet
temperature of the shell even though the baffle inclination is increased from 0° to 20°. However the shell-side pressure drop is decreased with increase in baffle inclination angle i.e., as the inclination angle is increased from 0° to 20°. The pressure drop is decreased by 4 %, for heat exchanger with 10° baffle inclination angle and by 26 % for heat exchanger with 20° baffle inclination compared to 0° baffle inclination heat exchanger. Hence it can be observed that shell and tube heat exchanger with 20° baffle inclination angle results in a reasonable pressure drop. Hence it can be concluded shell and tube heat exchanger with 20° baffle inclination angle results in better performance compared to 10° and 0° inclination angles.

Figure 2. Cut section showing velocity path lines along the shell for 0° baffle inclination angle for 1kg/s

Figure 3. Cut section showing velocity path lines along the shell for 10° baffle inclination angle for 1kg/s
4. Conclusions

The shell side of a small shell-and-tube heat exchanger is modeled with sufficient detail to resolve the flow and temperature fields. Following conclusions are drawn from the present study:

1. 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.
2. The pressure drop is decreased by 4 %, for heat exchanger with 10° baffle inclination angle and by 26 %, for heat exchanger with 20° baffle inclination angle.
3. The maximum baffle inclination angle can be 20°, if the angle is beyond 20°, the centre row of tubes are not supported. Hence the baffle cannot be used effectively.
4. Hence it can be concluded that shell-and-tube heat exchanger with 20° baffle inclination angle results in better performance compared to 10° and 0° inclination angles.

Nomenclature

- x, y, z: position coordinates, [-]
- u, v, w: velocity components, [ms⁻¹]
- do: tube outer diameter, [mm]
- q: Heat flux as a source term, [Wm⁻²]
- θ: baffle inclination angle, [degrees]
- B: central baffle spacing, [mm]
- Bc: baffle cut, [%]
- Di: shell inner diameter, [mm]
- L: heat exchanger length, [mm]
- Nb: number of baffles, [-]
- Nt: number of tubes, [-]
- T: temperature, [K]
- Vr: velocity vector, [-]

Greek symbols

- ρ: [Kgm⁻³]
- τ: [Nm⁻²]
- φ: Dissipation function [-]

References


[13]. Yong-Gang Lei, Ya-Ling He, Rui Li, Ya-Fu Gao, Effects of baffle inclination angle on flow and heat transfer of a heat exchanger with helical baffles, Chemical Engineering and Processing 47 (2008), pp. 2336–2345.

