Optimal design of porous baffle to improve the flow distribution in the tube-side inlet of a shell and tube heat exchanger

Kai Wang, Xin-Cheng Tu, Chan-Hyo Bae, Hyoung-Bum Kim

Research Center of Fluid Machinery Engineering and Technology, Jiangsu University, Zhenjiang 202013, PR China
Research Center for Aircraft Parts Technology, Gyeongsang National University, Jinju 660-701, Republic of Korea
School of Mechanical Engineering, Gyeongsang National University, Jinju 660-701, Republic of Korea
R & D Center, DongHwa Entec, Busan 618-230, Republic of Korea

ARTICLE INFO

Article history:
Received 4 March 2014
Received in revised form 22 September 2014
Accepted 27 September 2014

Keywords:
Shell and tube heat exchanger
Numerical optimization
Particle image velocimetry
Porous plate

ABSTRACT

An optimization study of a porous plate to improve the flow distribution in the tube side of a shell and tube heat exchanger was performed using numerical and experimental methods. The first assumption in the design stage of multichannel heat exchangers is that there is an even flow distribution along the multi-channels. However, this assumption is generally not upheld in real situations. An experimental study was first performed to confirm the effectiveness of a porous baffle in improving the flow distribution in the tube side of a shell and tube heat exchanger. The results showed that the baffle can largely decrease maldistribution. We then performed numerical optimization of the porous baffle shape. The optimized curvature of a porous plate and the distribution of circular holes in the baffle were determined using the numerical method. The optimization results showed that the area-weighted averaged absolute error of the flow rate of the optimized baffle decreased to one-third that of the prototype baffle model. For validation of the numerical optimization, the flow field with the optimized baffle was measured. Although there were some differences between the experimental and the numerical results, the results showed that the flow distribution using the optimized model was largely improved compared with that of the prototype porous baffle.

© 2014 Elsevier Ltd. All rights reserved.

1. Introduction

The even distribution of fluid flow passing through heat-transfer cores is a basic assumption in designing a heat exchanger. In real situations, the uniformity of flow distribution is hardly ever achieved due to geometric constraints or operating conditions. The maldistribution used in this study is defined as a deviation from this uniform flow distribution. Geometric-induced maldistribution (or mechanically induced maldistribution) is the most frequent causes of nonuniform flow. Geometric-induced maldistribution is related to the flow instability in the heat exchanger. This maldistribution generally results from the shape of the header/manifold/nozzle or the manufacturing tolerance of the shell and tube heat exchanger (STHX) and generates a gross change in the flow rate or passage to passage difference in the fluid flow. Self-induced maldistribution is caused by a change in the fluid properties, such as viscosity, density, or phase changes during the operation [1].

Although the flow maldistribution of the tube side is known to be less important in the performance of STHXs [2,3], Pacio and Dorao [4] reported a noticeable performance reduction in the evaporator of an STHX due to flow maldistribution of the tube-side inlet. They demonstrated that when the phase changes, the flow maldistribution has a large effect on the thermal performance of the heat exchanger. The LNG vaporizer, one of the core parts of LNG offshore plants, is a typical example of a STHX with phase transition [5].

Wen et al. [6,7] described the use of a porous baffle to improve the flow distribution in plate-fin heat exchangers. To the best of our knowledge, the optimization of the baffle shape and its influence on the tube-side inlet flow of an STHX have not yet been studied.

In this paper, we presented the optimized baffle model in the STHX which can dramatically improve the uniformity of the flow of the tube-side inlet. In addition to enhancing the performance of the vaporizer, we believe the results of this study provide new
insight into flow distribution control for use in designing heat exchangers.

2. Experimental method and apparatus

The flow field inside the header section was quantitatively measured using the particle image velocimetry (PIV) method. We measured the instantaneous velocity vector fields of the header without the baffle and with the prototype and numerically optimized the baffle.

Fig. 1(a) shows the experimental setup of the STHX flow loop system. The main components of the system are an STHX model, a flow meter, a pump, and a water reservoir. The STHX used in this study was a 1/3 scaled-down model of the prototype LNG vaporizer. Fully developed turbulent pipe flow condition was fulfilled at the inlet. The Reynolds number was based on the nozzle

![Diagram](image)

**Fig. 1.** Schematic diagram of experimental setup and apparatus.
diameter \(d\), and the mean axial velocity at the pipe was 26,000.
According to the Reynolds number, the flow rate was 100 LPM.
Although this value is smaller than that of the prototype STHX
(82,760), we conducted preliminary experiments of \(Re = 15,000\)
and 20,000 and confirmed that the mean flow characteristics did
not change and could be extrapolated. Fig. 1(b) shows the
coordinate system and geometries of the header and the location
of the inserted baffle. The baffle was located at \(x/d = -7\) where \(d\)
is the tube diameter.

The prototype baffle was designed based on knowledge of the
design of the plain baffles. A value of 30% was chosen for the porosity
of the baffle to avoid a separation bubble forming behind the
site of the baffle [8]. The circular holes in the baffle have a diameter
of 5, 8, and 10 mm along the radial direction. The center of the
baffle \((r = 20\text{ mm})\) has 0% porosity to enable it to function as an
impingement plate. In general, the distribution of the tube bundles
is not axisymmetric and there are more tubes in the planar plane
than in the perpendicular plane.

Two CCD cameras with a pixel resolution of \(1.6 \times 1.2\text{ K}\)
synchronized with an Nd:YAG laser were used to capture the flow
images. In total, 400 instantaneous velocity vector fields were measured and ensemble-averaged to obtain their mean properties.

### 3. Numerical optimization method

The design variables were the curvature of the baffle and the
geometries of the holes, including their diameter and distribution.
We used the automatic CFD optimization method and Isight \(\text{©}\) (Dassault Systems) integrated with Fluent \(\text{©}\) (Ansys Inc.) to determine
the effect of the geometries of the holes. The optimal Latin hyper-
cube scheme was used for this optimization process. The objective
function of the optimization process was to minimize the area-weighted averaged absolute error of the flow rate defined in
Eq. (1):

\[
\Delta Q_{w} = \frac{\sum_{i=1}^{n} z_{i} [(Q_{N}(r_{i}) - Q_{t}(r_{i}))/Q_{t}(r_{i})]}{n}
\]  

where \(r_{i}\) is the sample point along the radial direction \((0 \leq r_{i} \leq D/2)\),
\(Q_{N}(r_{i})\) is the numerical simulation value of the flow rate at the
\(r_{i}\) location, and \(Q_{t}(r_{i})\) is the theoretical value of the flow rate at the
\(r_{i}\) location. In the original averaged absolute error of the flow rate
proposed by Wen and Li [6], the flow rate of each channel of the
plate-in heat exchanger was used. However, as we used the flow
rate calculated from the inlet axial velocity profile instead of the

---

**Fig. 2.** Mean velocity fields of tube-side of the STHX model from PIV measurements.
flow rate of each tube, we used $a_i$ to compensate for the difference between Wen and Li's [6] definition and the definition employed in the current study. The definition of $a_i$ is the area ratio of $A_i (=2\pi r_i \times dr)/A_{in}$, where $A_{in}(=\pi D^2/4)$ is the area of the tube-side inlet. The theoretical value is based on the assumption that the axial velocity has a uniform distribution along the radial direction. In this study, the theoretical axial velocity in front of the tube inlet was 2.36 cm/s.

**Fig. 3.** Comparisons of mean velocity and accumulated flow rate depending on the baffle at $x/d = -1.3$.

**Fig. 4.** Comparison of mean velocity between numerical and experimental results at $x/d = -1.3$.

**Fig. 5.** Comparison of area-weighted averaged absolute error of flow rate according to the different curvature.
4. Results

4.1. Effect of prototype baffle on flow maldistribution

Fig. 2 shows the mean velocity fields of the tube-side inlet of the STHX model with and without the prototype baffle based on the PIV measurement results. Due to the joint blocking laser light illumination, velocity information on this blocked region is missing. Fig. 2(a) and (b) presents the perpendicular and planar planes without the baffle. The figures clearly show that the strong jet from the inlet nozzle directly impinges the tube bundle. A large circulation area is apparent at the corner of the tube and at the side of the outer wall. Due to the asymmetric distribution of the tube bundles, the location of the vortex core is in a different position in the perpendicular and planar planes. Although the radial position of the vortex core is similar in both the perpendicular and planar planes, the vortex core in the perpendicular plane is closer to the tube inlet area than in the planar plane.

One distinctive finding was the axial velocity at the uppermost tube of the perpendicular tube. This had a negative value of 2.25 cm/s, whereas that of the outermost tube of the planar plane was nearly zero (0.25 cm/s). Thus, the outer side of the tube bundle has a zero or negative mass flow in this experimental model. Consequently, we can expect a reduction in the heat performance of the STHX. Another study also reported a negative or zero velocity at the channels of a compact heat exchanger [9].

Fig. 2(c) and (d) presents the mean velocity fields of the header region with the prototype baffle. As can be seen, the vortex located at the corner of the tube inlet decreased, and the location of vortex core in the perpendicular and planar planes has the same axial and radial positions. Thus, the porous baffle suppressed the corner vortex flow. Counter rotating vortices appeared behind the impingement region, and small vortices appeared behind the porous baffle. We think that this caused the additional head loss.

To quantitatively investigate the improvement in the flow distribution due to the prototype baffle, we extracted the line data at the location of \( x/d = -1.3 \) from the field results (Fig. 3). As the
installation of the prototype baffle causes the flow behind the baffle to be symmetric, we present the averaged results from the perpendicular and planar plane data.

In Fig. 3, the black solid line represents the ideal uniform axial velocity profile of 100 LPM. Fig. 3(a) and (b) shows a comparison of the axial and radial velocity, and Fig. 3(c) displays the accumulated flow rate calculated using the axial velocity. The figure clearly shows the strong momentum at the impingement region and the negative axial velocity at the outer radius region in the without-the-porous-baffle (WOPB) model. On the other hand, the axial velocity in the with-the-porous-baffle (WPB) model converged to the ideal axial velocity profile. It can be seen that both the magnitude of axial velocity in the center region and the axial velocity in the outermost region decreased. The radial velocity profiles of the WOPB and WPB models exhibited a large difference. In the WOPB model, the local maximum was at \( r = 0.04 \) m, which is the same as that of the inlet nozzle radius but generally larger than that in the WPB model, except in the near wall region \( (r > 0.11 \) m). The negative radial velocity region in the WPB model was due to the low pressure of the vortex that formed behind the impingement region.

In addition, although the axial and radial velocity profiles of the WPB model showed various local maximum locations along the radial direction, the accumulated flow rate results shown in Fig. 3(c) demonstrate that the porous baffle improved the flow uniformity in the tube side of the STHX.

The comparison of the mean velocities and flow rates revealed that the prototype baffle can dramatically improve the flow uniformity compared with the conventional vaporizer without a porous baffle. This result was similar with other researches [6,7].

4.2. Numerical optimization results of porous baffle curvature

To increase the flow uniformity and reduce the head loss, numerical optimization was performed. To quantitatively compare the difference between the numerical simulation and the PIV measurements, a comparison of the mean velocity and the accumulate flow rate at \( x/d = -1.3 \) is shown in Fig. 4. The profile of the flow rate and the mean velocity is similar, except for a slight difference at the wall region. The maximum difference (16%) in the flow rate appeared at the \( r = 0.13 \) m location. The CFD has a smaller gradient and larger uniformity and shows weak vortices at the wall region compared with the experimental results. This suggests that the CFD analysis overestimated the effect of the porous baffle compared with the experimental analysis. Regardless of this discrepancy...
between the CFD and experimental results, we believe the numerical approach presented herein can help to guide optimized baffle design.

4.2.1. Numerical optimization of baffle curvature

The prototype baffle has a straight shape, and we investigated the ability of baffle curvature to improve the flow uniformity. We can see that the curvature of the baffle has a large effect on the flow rate. The area-weighted averaged absolute error of the flow rate is shown in Fig. 5. The \( \Delta Q_w \) at \( R/D = -1 \) has a minimum value of 1.13%, which is a 39.9% decrease of the error (1.88%) of the prototype baffle. In general, a porous baffle with a convex shape had a smaller error than a baffle with a concave shape.

The formation of a small vortex at the impingement region, which is the center of the baffle, appeared. To avoid reversal flow, we placed circular holes with diameters of 4 mm to provide 30% porosity. The results showed the vortex formation regions have disappeared, and the averaged absolute error of the flow rate under the conditions in the figure are 1.89%, which is similar to that of the curvature baffle with the impingement zone. Thus, the impingement zone does not necessarily contribute to the reduction of the flow maldistribution of the STHXs. We conducted an optimization study of the circular holes in the baffle with the optimum curvature.

4.2.2. Numerical optimization results for the distribution of the holes in the baffle

The initial values of the design variables related to the distribution of the holes are shown in Fig. 6(a). The design variables were as follows: \( D_{01}, D_{02} \in [3, 5], D_{03}, D_{04}, D_{05} \in [4, 6], D_{06}, D_{07}, D_{08} \in [6, 10], D_{09}, D_{10}, D_{11} \in [8, 10], L_j = [L_{j-1}, L_{j+1}], j = 01, 02, \ldots, 10. \)

After the automated optimization process, we obtained the minimum of the averaged absolute error of the flow rate (0.65%). This is about one-third that of the WPB model. The detailed design values and the accumulated flow rate comparison are presented in Fig. 6(b). The figure shows that the curvature-and-hole optimized model is the closest to the theoretical profile.

4.3. Experimental validation of numerical optimization

We made the optimized baffle based on the numerical optimization results and performed a validation study under the same experimental condition. Fig. 7 shows the velocity results of the tube-side inlet with the optimized baffle. Compared with Fig. 2(c) and (d), the distinctive feature of the flow structure in the tube-side inlet is the reduction in the corner vortex and the disappearance of the central vortex formation region.

Due to this suppression of the vortex, the streamline became straighter at the inlet of the tube side. The line velocity and accumulated flow rate at \( x/d = -1.3 \) was compared and is presented in Fig. 8. This figure clearly shows the change in the axial and radial velocity profile due to the baffle optimization. The axial velocities of the optimized baffle at the center region increased compared to that of the baffle in the prototype model following the removal of the impingement area. Except for this center region, the optimized baffle has a flatter axial velocity profile and smaller negative values at the near wall region. The radial velocity comparison in Fig. 8(b) also shows the flatter profile and larger reduction of magnitude at the region closest to the wall. Another distinctive feature of the optimized baffle was the disappearance of the oscillation of the axial and radial velocity components. Due to the step change in the size of the diameters of the holes, the prototype baffle showed several local maximum and minimum values of velocity components along the radial direction. The optimized baffle does not exhibit this characteristic and has fewer variations in velocity profile.

5. Conclusions

In an attempt to improve the flow uniformity and the heat performance of an LNG vaporizer, we investigated the effect of a
porous baffle in front of the tube side. From velocity measurements, we quantitatively confirmed that the porous baffle dramatically improved the uniformity of the flow. We then performed a numerical study to find the optimized porous baffle with a minimum averaged absolute error of flow rate. Based on the results, we proposed the best baffle shape and the optimum distribution of holes at the baffle. After the numerical optimization, we carried out an experimental validation study of the proposed baffle. Although there were some discrepancies between the PIV measurements and the CFD results, the results showed that the flow uniformity was improved using this optimized model. The optimization results confirmed that the suppression of the corner vortex is important to improve the flow uniformity and the role of impingement zone in the porous baffle is less important for reducing the flow maldistribution. The optimization of the hole geometry removed small vortices behind the baffle and the fluctuation of the axial and radial velocity along the radial direction. We believe these data will be very useful for the design of STHXs, such as LNG vaporizers.

Conflict of interest
None declared.

Acknowledgments
The work was supported from the Priority Research Centers Program (2012-048078), Basic Science Research Program (2012-008918) through the National Research Foundation of Korea and a grant (08GASPLANTC04) from the KAIA program of the Republic of Korea. One of the authors (Wang Kai) was supported by the National Natural Science Foundation of China (Grant Nos. 51179075, 51109095, 51239005 and 51209105), the National Science & Technology Pillar Program of China (Grant Nos. 2011BAF14B03 and 2013BAK06B02) and Senior Professional Scientific Research Project of Jiangsu University of China (Grant No. 12JDG045).

References