Influence of the louver and delta winglet geometry on the thermal hydraulic performance of a compound heat exchanger

H. Huisseune, C. T'Joen, P. De Jaeger, B. Ameel, S. De Schampheleire, M. De Paepe

A B S T R A C T

Louvered fin heat exchangers are frequently used in heat transfer applications where air is one of the working fluids. By punching delta winglet vortex generators in the louvered fins, the size of the tube wakes can be reduced. The objective of this paper is to study the influence of the louver and delta winglet geometry on the thermal and hydraulic performance of such a compound heat exchanger. To this end, three-dimensional numerical simulations were performed. The effect of the louver and delta winglet geometry on the heat exchanger’s performance results in conflicting design requirements. A small fin pitch and large louver angle cause a strong flow deflection and thus a large contribution of the louver. But in this case the generation of longitudinal vortices is suppressed and thus the effect of the delta winglets is small. Also the delta winglet geometry itself has an important influence. A well-considered geometry and location of the delta winglets is essential for an improved performance. Delta winglets completely located in the tube wakes should be avoided, because in this case they do not cause any enhancement effect.

1. Introduction

When exchanging heat with air, the main thermal resistance is located at the air side of the heat exchanger (75% or more [1]). To improve the heat transfer rate, the air side heat transfer surface area can be enlarged by adding fins. When a high compactness is desirable, complex interrupted fin surfaces are preferred, because they prevent the formation of thick boundary layers and promote unsteadiness. Fig. 1a represents the interrupted section of a louvered fin surface between the tubes. Louvered fins are frequently used in air conditioning devices, heat pumps, car radiators, refrigeration, etc. This fin type consists of an array of flat plates (the louvers) set at an angle to the incoming flow. The characteristic parameters of the louvered fin geometry are also indicated in Fig. 1a. Numerous studies have been performed on this fin design, focussing on the flow deflection [2–4], onset of unsteadiness [4–6], thermal wakes behind the louvers [7,8], etc. Flow deflection is characteristic for louvered fin arrays. Through a finite-difference analysis, Achiaia and Cowell [2] showed that increasing the Reynolds number results in a transition in the flow behavior from duct-directed to more louver-directed flow (see Fig. 1b). This is an example of ‘boundary layer driven flow’. At low Reynolds numbers the thick boundary layers block the passage between the louvers, forcing the flow to go straight through. As the Reynolds number increases, the boundary layers become thinner and the passage opens up, aligning the flow with the louvers and extending the flow path. This results in an increased heat transfer rate. But as the flow path is extended, the frictional pressure drop also increases. The degree to which the flow follows the louvers is quantified by the flow efficiency. Next to the Reynolds number, the flow efficiency is also strongly dependent on the louvered geometry. This is shown by Zhang and Tafti [3] through numerical simulations. These numerical findings were in good agreement with the flow visualization experiments of DeJong and Jacobi [4].

The interrupted section of Fig. 1 needs to be connected to the tubes to form the heat exchanger. In modern heat exchangers this is done through a transition of the angled louvers to a flat fin surface (the so called landing), which is then connected to the tubes through mechanical or hydraulic expansion [9].

The major drawback of the interrupted fin designs is that their associated pressure drop is significant. In contrast to interrupted fin patterns, plain fins with longitudinal vortex generators enhance the heat transfer rate with relatively low penalty of the pressure drop [10]. The generated vortices provide a swirling motion to the flow field which causes an intense mixing of the main flow with the flow in the wall regions. They also reduce the thickness of the thermal boundary layers and encourage flow destabilization. This results in an enhanced heat transfer. Among the different
types of longitudinal vortex generators, delta winglet pairs are very attractive as enhancement technique, when taking heat transfer as well as pressure drop into account \[11\]. An example is shown in Fig. 2a. When delta winglets are used in fin and tube heat exchangers, they can reduce the poor heat transfer region in the tube wakes. To this purpose, the common-flow-down configuration, shown in Fig. 2b, is frequently used. The delta winglet pair is placed downstream the tube in the near wake region in order to introduce high momentum fluid behind the tube and improve the poor heat transfer in the wake region \[12\].

Tiwari et al. \[14\] studied a single tube in a rectangular duct with delta winglet vortex generators in a common-flow-down configuration. Numerical simulations were carried out for isothermal fins and laminar flow conditions. Increased spanwise Nusselt numbers

---

**Nomenclature**

- $A_c$: minimum cross sectional flow area (m$^2$)
- $A_o$: external heat transfer surface area (m$^2$)
- $A_f$: fin surface area (m$^2$)
- $c_p$: specific heat capacity (J/kg K)
- $D_h$: hydraulic diameter (Eq. (1)) (m)
- $D_o$: outer tube diameter (m)
- $f$: friction factor (Eq. (7)) (–)
- $F$: pumping power factor according to LaHaye et al. \[51\] (Eq. (10)) (–)
- $F_n$: pumping power factor according to Soland et al. \[53\] (Eq. (12)) (–)
- $F_p$: fin pitch (m)
- $G_c$: mass flux in the minimum cross sectional flow area (kg/m$^2$ s)
- $h^*$: height ratio of the delta winglet (–)
- $h_o$: external convection coefficient (W/m$^2$ K)
- $j$: Colburn $j$-factor (Eq. (6)) (–)
- $J$: heat transfer performance factor according to LaHaye et al. \[51\] (Eq. (9)) (–)
- $J_n$: heat transfer performance factor according to Soland et al. \[53\] (Eq. (11)) (–)
- $L$: flow depth (m)
- $L_p$: louver pitch (m)
- $m$: mass flow rate (kg/s)
- $\Delta P$: pressure drop (Pa)
- $Pr$: Prandtl number (–)
- $Q$: heat transfer rate (W)
- $Re_{Lp}$: Reynolds number based on the louver pitch and the velocity in the minimal cross sectional flow area (–)
- $s$: fin spacing = difference between fin pitch and fin thickness (m)
- $T$: temperature (K)
- $t_f$: fin thickness (m)
- $V_c$: velocity in the minimal cross sectional flow area (m/s)
- $V_{\infty}$: free-stream velocity (m/s)
- $P_l$: transversal tube pitch (m)
- $\Delta x$: streamwise delta winglet position (m)
- $y_p$: distance to the fin surface from the adjacent cell centroid (m)
- $\Delta z$: spanwise delta winglet position (m)

**Greek symbols**

- $\alpha$: angle of attack of the delta winglet vortex generator (°)
- $\beta$: surface efficiency (–)
- $\eta_f$: fin efficiency (–)
- $\lambda$: thermal conductivity (W/m K)
- $\Lambda$: delta winglet aspect ratio (–)
- $\nu$: kinematic viscosity (m$^2$/s)
- $\vartheta$: louver angle (°)
- $\rho$: density (kg/m$^3$)
- $\sigma$: contraction ratio (i.e. the ratio of the minimal cross sectional flow area to the frontal area) (–)
upstream and downstream of the tube were found, due to the formation of horseshoe vortices and a reduction of the wake zone, respectively. Pressure drop results were not reported. Fiebig et al. [15] tested inline and staggered tube bundles consisting of three tube rows and delta winglets in common-flow-down orientation behind each tube. For the inline arrangement the heat transfer increased by 55–65% and the friction factors increased by 20–45% for the range of Reynolds numbers from 600 to 2700 (based on the inlet velocity and two times the channel height). For the staggered arrangement a heat transfer augmentation of 9% was found accompanied by a 3% increase in friction factor for the same Reynolds number range. The optimal common-flow-down position of the delta winglet pair was experimentally determined by Pesteei et al. [16]. The best thermal hydraulic performance was found for delta winglets located at $\Delta x = 0.5D$ and $\Delta y = 0.5D$ ($D$ is the outer tube diameter and $\Delta x$ and $\Delta y$ are, respectively the streamwise and spanwise distance between the tube center and the point where the leading edge of the winglet intersects with the fin surface). Increasing the angle of attack results in better heat transfer because stronger vortices are produced which enhance the fluid mixing. Unfortunately, also the flow resistance (and thus the pressure loss) increases with the angle of attack. Fiebig et al. [17] found that the best heat transfer performance is achieved for an angle of attack equal to 45°.

Heat exchanger manufacturers are continuously searching for new and better designs. The next generation of enhanced fin surfaces will combine known enhancement techniques, resulting in so-called compound heat exchangers [18]. The aim is that the compound design results in a higher performance than the individual techniques operating separately – is believed to have a wide applicability. However, only a few studies on this kind of compound design were found in literature and they all focused on flat tube heat exchangers (typically for automotive applications). In this work round tube heat exchangers combining louvered fins with delta winglet pairs in a common-flow-down orientation are studied. In a previous study the local flow structures which affect the heat transfer and pressure drop in the considered compound heat exchanger were analyzed for a single geometry [25]. The objective of this paper is to investigate the influence of the most important geometrical parameters on the heat transfer and friction characteristics of the compound heat exchanger. To this end, three-dimensional numerical simulations were performed.

2. Three-dimensional computational domain

The three-dimensional computational domain of the louvered fin geometry with vortex generators is shown in Fig. 3. Three tube
rows in a staggered arrangement are considered. Delta winglet vortex generators are punched out of the fin surface in a common-flow-down arrangement behind each tube. Each louver element between the tubes consists of an inlet louver, an exit louver and two louvers on either side of the turnaround louver. Each louver transitions from an angle \( \alpha \) into a flat landing adjoining the tube surface. The spanwise dimensions of the flat landing and transition part were chosen as in [26,27], i.e. \( 0.25D_0 \) for the minimum flat landing (between the turnaround louver and tube) and \( 0.5D_0 \) for the transition part. Periodic conditions are applied on both sides of the domain as well as on the top and bottom. The height of the computational domain is equal to the fin pitch \( F_p \) and the width is equal to transversal tube pitch \( P_t \). The geometry is located in the middle with half a fin spacing above the fin surface and half a fin spacing below. The entrance length upstream of the fin equals 5 times the fin pitch \( F_p \) and the domain extends 7 times the tube diameter \( D_0 \) downstream of the fin, as was suggested by Jang et al. [28].

A reference geometry was defined and one geometrical parameter was changed at a time. In total the variation of five geometrical parameters are fixed with values given in Table 2. The other geometrical parameters are fixed with values given in Table 2. The values are selected based on a literature review of louvered fins and plain fins with delta winglets.

### 3. Computational method

The mesh was generated using Gambit\textsuperscript{®}. The air domain as well as the solid fin material were meshed to take the fin conduction into account. The quality of the mesh was carefully assessed during the meshing. The computational domain was divided into several subdomains. The fin material was meshed with quad elements (three cells over the fin thickness). Most of the air subdomains were also meshed with quad elements. Only the subdomains with the transition zone between the angled louver and flat landing and the subdomains surrounding the delta winglets were meshed using unstructured tetrahedral elements. The mesh gets finer towards the fin surface to capture the temperature and velocity gradients in the near wall regions.

The mass, momentum and energy balance equations were solved using the CFD software ANSYS Fluent\textsuperscript{©} 12.0.16. The flow is assumed to be laminar. This assumption is reasonable for the considered Reynolds numbers \( (Re_{dp} = 220–915) \) as the experimental observations of Antoniou et al. [32] illustrated that the flow in louvered fin arrays is laminar for Reynolds numbers \( Re_{dp} \) up to 220–915.

### Table 2

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Symbol</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Outer tube diameter</td>
<td>( D_0 ) (mm)</td>
<td>6.75</td>
</tr>
<tr>
<td>Fin thickness</td>
<td>( t_f ) (mm)</td>
<td>0.12</td>
</tr>
<tr>
<td>Louver pitch</td>
<td>( L_p ) (mm)</td>
<td>1.5</td>
</tr>
<tr>
<td>Transversal tube pitch</td>
<td>( P_t ) (mm)</td>
<td>17.6</td>
</tr>
<tr>
<td>Longitudinal tube pitch</td>
<td>( P_l ) (mm)</td>
<td>13.6</td>
</tr>
<tr>
<td>Streamwise DW position</td>
<td>( \Delta x ) (mm)</td>
<td>0.5( D_0 )</td>
</tr>
<tr>
<td>Spanwise DW position</td>
<td>( \Delta z ) (mm)</td>
<td>0.3( D_0 )</td>
</tr>
</tbody>
</table>

### Table 1

<table>
<thead>
<tr>
<th>Case No.</th>
<th>( F_p ) (mm)</th>
<th>( \theta ) (°)</th>
<th>( \alpha ) (°)</th>
<th>( h^* )</th>
<th>( A )</th>
<th>Parameter</th>
</tr>
</thead>
<tbody>
<tr>
<td>Reference</td>
<td>1.71</td>
<td>28</td>
<td>35</td>
<td>0.9</td>
<td>2.0</td>
<td></td>
</tr>
<tr>
<td>1</td>
<td>1.20</td>
<td>28</td>
<td>35</td>
<td>0.9</td>
<td>2.0</td>
<td>( F_p )</td>
</tr>
<tr>
<td>2</td>
<td>1.99</td>
<td>28</td>
<td>35</td>
<td>0.9</td>
<td>2.0</td>
<td>( \theta )</td>
</tr>
<tr>
<td>3</td>
<td>1.71</td>
<td>22</td>
<td>35</td>
<td>0.9</td>
<td>2.0</td>
<td>( \alpha )</td>
</tr>
<tr>
<td>4</td>
<td>1.71</td>
<td>35</td>
<td>35</td>
<td>0.9</td>
<td>2.0</td>
<td>( \alpha )</td>
</tr>
<tr>
<td>5</td>
<td>1.71</td>
<td>28</td>
<td>30</td>
<td>0.9</td>
<td>2.0</td>
<td>( \alpha )</td>
</tr>
<tr>
<td>6</td>
<td>1.71</td>
<td>28</td>
<td>25</td>
<td>0.9</td>
<td>2.0</td>
<td>( \alpha )</td>
</tr>
<tr>
<td>7</td>
<td>1.71</td>
<td>28</td>
<td>35</td>
<td>0.5</td>
<td>2.0</td>
<td>( h^* )</td>
</tr>
<tr>
<td>8</td>
<td>1.71</td>
<td>28</td>
<td>35</td>
<td>0.9</td>
<td>1.5</td>
<td>( A )</td>
</tr>
</tbody>
</table>

Fig. 4. Comparison of the fine and coarse mesh resolution: (a) Colburn \( j \)-factors and (b) friction factors.
approximately 1300. Other authors also used the laminar flow assumption for their heat exchanger simulations, often verified against experimental data [28,33–36].

At the inlet a uniform velocity parallel to the fin was imposed and the air inlet temperature was set to 20 °C. At the outlet the static pressure was set to 0 Pa (pressure outlet boundary condition). This is also indicated in Fig. 3a. The tube wall thickness was not meshed, but the shell conduction approach was used [37]. A tube wall thickness of 0.27 mm was specified. When heat is exchanged with air, the main thermal resistance is located at the air side. Consequently, the tube wall temperature is mainly determined by the temperature of the fluid flowing through the tubes. For evaporators and condensers, the fluid (refrigerant) temperature remains nearly constant at the saturation temperature throughout the heat exchanger. For other heat exchangers (e.g., the cooling of water), the fluid temperature typically varies about 4 °C per meter tube [38]. Due to the high thermal conductivity of the tube material (λ = 400 W/m K for copper) and the small change in fluid temperature which occurs on the tube side over the length of one fin pitch, a constant tube wall temperature can be assumed. In the current simulations a constant tube wall temperature of 50 °C was applied in the three tube rows. No slip boundary conditions were applied on the tube and fin surfaces. The double precision segregated solver was used to solve the standard Navier–Stokes equations. The energy equation was turned onto compute the heat transfer through the tube walls and fin material and in the air. The SIMPLE algorithm was applied for the pressure–velocity coupling. The discretization of the convective terms in the governing equations was done via a second order upwind scheme, while a second order central differencing scheme was applied for the diffusive terms. The gradients were evaluated via the least squares cell based method. The pressure gradient in the momentum equations was treated via a second order discretization scheme. Convergence criteria were set to 10⁻⁸ for continuity, velocity components and energy. Setting lower values for these criteria did not result in any notable differences in the flow field and heat transfer predictions. For the air properties the same settings were used as by T’Joen et al. [39] and Perrotin and Clodic [36]: the air density was calculated as for an incompressible ideal gas, the specific heat and thermal conductivity were set to constant values (c_p = 1006 J/kg K and λ = 0.02637 W/m K) and the dynamic viscosity was calculated with the Sutherland approximation. The properties of the aluminum fin and tubes are assumed to be constant: 

\[
p = 2719 \text{ kg/m}^3, \quad \lambda = 202.4 \text{ W/m K} \quad \text{and} \quad c_p = 871 \text{ J/kg K}.
\]

Only for the smallest Reynolds numbers (Re_{fin} < 200) steady simulations were found to converge. The Reynolds number Re_{fin} is based on the velocity V_f in the minimum cross sectional flow area and the hydraulic diameter D_h:

\[
D_h = \frac{4 A_L}{A_o}
\]

with A_L the minimum cross sectional flow area, L the flow depth and A_o the total heat transfer surface area. For Re_{fin} > 200 unsteady simulations were performed (first order accurate in time). The time step was selected based on the characteristic Strouhal number for louver unsteadiness and tube wake instabilities. Tafti and Zhang [6] reported Strouhal numbers between 0.3 and 0.5 for louver instabilities. For the tube wake instabilities a typical Strouhal number of 0.2 was used [40,41]. For all simulations, the time step varied between 1 μs and 1 ms (dependent on the Reynolds number). This resulted in at least 25 calculations per shedding period. It also allowed the residuals to decrease below 10⁻⁸ in less than 50 iterations per time step. The mass-weighted average pressure drop and outlet temperature were monitored during the iterations to determine if the simulations were converged. Local temperatures in the tube and louver wakes were also monitored. Once convergence was reached, these temperatures varied in function of the time around a mean value. The data of the unsteady simulations reported in this paper are the time averaged values. The averaging is done over the time interval an air particle needs to travel about three times the length of the computational domain and after the flow has reached a statistically fully developed state. This corresponds to at least 10 shedding periods for the tube wake instabilities and at least 80 shedding periods for the louver wake instabilities. Averaging over a longer time interval did not result in any notable differences: averaging out over 3, 9 and 27 times the length of the computational domain resulted in differences smaller than 0.05% for the highest Reynolds number. For each simulation the energy balance was checked: the net heat transfer rate between the air inlet and outlet differed less than 0.01% from the total heat transfer rate of the tube and fin surface.

All simulations were performed on two six core Intel Xeon X5680 processors with 48 Gb RAM and 3.33 GHz CPUs. The average computational time for one case was about 3 days. For the highest Reynolds numbers the computational time reached up to 6 days.

4. Data reduction method

The heat transfer rate Q at the air side was determined as:

\[
Q = m_{air} c_{p,air} (T_{air,out} - T_{air,in})
\]

with c_p the specific heat capacity evaluated at the mean temperature of the inlet and outlet temperatures. The air side convection coefficient was calculated using the LMTD (logarithmic mean temperature difference) method:
ho is the total exterior heat transfer surface area and \( \eta_o \) is the surface efficiency. Due to the fixed wall temperature the correction factor \( F \) is equal to unity [42]. The logarithmic mean temperature difference is expressed as:

\[
\text{LMTD} = \frac{(T_{\text{wall}} - T_{\text{air, out}}) - (T_{\text{wall}} - T_{\text{air, in}})}{\ln\left(\frac{T_{\text{wall}} - T_{\text{air, out}}}{T_{\text{wall}} - T_{\text{air, in}}}\right)}
\] (4)

The surface efficiency \( \eta_o \) was calculated with the fin efficiency \( \eta_f \):

\[
\eta_o = 1 - \frac{A_f}{A_o} (1 - \eta_f)
\] (5)

\( A_f \) is the fin surface. The fin efficiency \( \eta_f \) was determined using the equivalent circular fin method of Schmidt [43], as was also done by many other authors [9,44,45]. Because the fin efficiency is dependent on the air side convection coefficient \( h_o \), it resulted from iterative calculations. The air side convection coefficient is represented dimensionless as the Colburn \( j \)-factor:

\[
j = \frac{h_o}{\rho \cdot c_p \cdot \sqrt{Pr}} \cdot \frac{Pr}{V_c}
\] (6)

\( V_c \) is the velocity in the minimum cross sectional flow area and \( Pr \) is the Prandtl number. The pressure drop across the heat exchanger is presented as the Fanning friction factor. This friction factor is calculated as proposed by Kays and London [46]:

\[
f = \frac{A_c}{A_o} \rho \frac{\Delta p}{\rho \cdot V_{\infty} \cdot C_f} \left( 1 + \sigma - \frac{\rho_{\infty} \cdot \Delta p}{\rho_{\infty} \cdot \Delta p_{\text{out}}} \right)
\] (7)

\( G_c \) is the mass flux in the minimal cross sectional flow area \( A_c \). The contraction ratio \( \sigma \) is defined as the ratio of \( A_c \) to the frontal area. The friction factor includes the entrance and exit pressure loss.

5. Validation of the numerical results

The computational domain should be meshed in such a way that the mesh is fine enough to result in accurate predictions, but also not too fine as this results in a too long computational time. The mesh spacing near the walls has a significant influence on the accuracy of the computed heat transfer coefficients. In laminar flows the grid adjacent to the wall should obey [37]:

\[
y_p \cdot \sqrt{\frac{V_{\infty}}{v \cdot x}} \leq 1
\] (8)

Here \( y_p \) is the distance to the fin surface from the adjacent cell centroid, \( V_{\infty} \) is the free-stream velocity, \( v \) is the kinematic viscosity of air and \( x \) is the distance along the fin surface from the starting point of the boundary layer. Eq. (8) is based upon the Blasius solution for laminar flow over a flat plate at zero incidence [37].

Fig. 6. Influence of the fin pitch \( F_p \) as function of the inlet velocity \( V_{\infty} \): (a) the Colburn \( j \)-factor, (b) the friction factor, (c) the area goodness factor and (d) the modified volume goodness factor.
A grid independency study was performed for the compound heat exchanger corresponding to case 4 in Table 1. Two mesh resolutions were evaluated. For the fine mesh \( y_p \) was estimated in a conservative way. The shortest surfaces for the simulated geometry are the louvers (streamwise length \( L_p \)). To have cells which are small enough for 95% of the boundary layer length, \( x \) was evaluated as 0.05\( L_p \). The highest inlet velocity used in the calculations is 5.25 m/s. The maximum cross sectional averaged velocity in the heat exchanger is then 5.25/\( \sigma \) (with \( \sigma \) the contraction ratio). This velocity was chosen as upper limit of the free-stream velocity \( V_{\text{in}} \). The height of the boundary layer cells adjacent to the fin surface for the fine mesh is then 2\( y_p \) = 22 \( \mu \)m. The aspect ratio of these cells was set to 2. The boundary layers consist of 4 cell rows with a growth rate equal to 1.3. The maximum cell size in the heat exchanger domain is 0.08 mm. This fine mesh counts about 14,000,000 cells. As for this mesh resolution \( y_p \) was estimated in a conservative way, a second coarser mesh was examined. The height of the boundary layer cells adjacent to the fin surface was set to 40 \( \mu \)m. This corresponds to the same cell size as used in the fin material (=\( t_f / 3 \)). The aspect ratio of the boundary layer cells is 2.5 and the growth rate is 1.3. The boundary layer for this coarser mesh consists of 3 cell rows. The maximum cell size in the heat exchanger domain is 0.10 mm. The coarse mesh counts about 4,300,000 cells.

The Colburn and friction factors are calculated for four inlet velocities (\( V_{\text{in}} = 0.63, 1.26, 2.69 \) and 5.25 m/s). In Fig. 4 the results are plotted for both mesh resolutions. The differences between the coarse and fine mesh simulations are very small for the thermal as well as the hydraulic characteristics. Together with the computing resource and time considerations, this justifies the use of the coarse mesh for the simulations. As the maximum cell size of the coarse mesh is 0.10 mm, it is much finer than the mesh used by Perron and Clocic [36] (their average cell size equals the fin thickness) and Atkinson et al. [47] (their minimum cell size equals the fin thickness) for their three-dimensional simulations. Both studies reported a good to very good agreement between the experimental data and the 3D numerical predictions of pressure drop and heat transfer in louvered fin flat tube heat exchangers.

The numerical results were also validated against experimental data. A wind tunnel experiment was set up and a compound heat exchanger was tested. The same geometry was also simulated using CFD. The Colburn \( j \)-factors and friction factors were compared. A description of the experimental equipment and procedure, the data reduction method and the uncertainty analysis is presented in Huisseune [48]. The results are plotted in Fig. 5. The open symbols represent the wind tunnel measurements and the filled symbols are the simulation results. The uncertainty bars are indicated on the experimental data. As explained in Huisseune [48], the slight degree of turbulence inherent to the wind tunnel setup might explain the differences between the experimental data and the numerical predictions. In general, there is an acceptable match between the simulations and the experiments within the considered uncertainty ranges (the mean deviation of the Colburn \( j \)-factors is 6.2% and of the friction factors 5.5%).

6. Results and discussion

The influence of five geometrical parameters is investigated: the louver pitch \( F_p \), the louver angle \( \theta \), the delta winglet height ratio \( h \) and the delta winglet aspect ratio \( A \).

6.1. Variation of the fin pitch \( F_p \)

The effect of the fin pitch on the Colburn \( j \)-factor and the friction factor is shown in Fig. 6a and b, respectively. The results for three different fin pitches (\( F_p = 1.20, 1.71 \) and 1.99 mm) are plotted as function of the inlet velocity \( V_{\text{in}} \). The Colburn \( j \)-factor and the friction factor decrease with the inlet velocity. For the smallest fin pitch \( F_p = 1.20 \) mm, the slope of the Colburn \( j \)-characteristic decreases with reducing velocities below 1.2 m/s. This is because at low inlet velocities the heat exchanger with the smallest fin pitch \( F_p = 1.20 \) mm is oversized. In other words, the air reaches the tube wall temperature before the end of the heat exchanger. Consequently, part of the heat exchanger does not contribute to the heat transfer, but only causes pressure drop. For \( V_{\text{in}} > 1.2 \) m/s the highest Colburn \( j \)-factors are found for the smallest fin pitch (\( F_p = 1.20 \) mm). However, the largest fin pitch (\( F_p = 1.99 \) mm) does not correspond with the lowest Colburn \( j \)-factors. This is because different enhancement effects interact. First, for large fin pitches, the flow development length is long. On average, the boundary layers are thinner than for small fin pitches and thus the overall Colburn \( j \)-factors are higher. Second, consider the louver and delta winglet geometry. Zhang and Tafti [3] and DeJong and Jacobi [4] showed that in louvered geometries the thermal performance is enhanced through alignment of the flow with the louvers. From a thermal point of view a smaller fin pitch is preferred as this results in a more louver directed flow. The delta winglet geometry, on the other hand, enhances the thermal performance because the generated longitudinal vortices reduce the size of the tube wakes, thin the boundary layers and cause a better fluid mixing. In contrast to the louver geometry, a too small fin pitch is not preferred because then the generation of vortices is suppressed. The vortices
do also not propagate far downstream as they are destroyed by the deflected flow in the downstream tube row. This is illustrated in Fig. 7. Here the X vorticity is plotted in a plane parallel to the inlet halfway between the punched delta holes in the first tube row and the inlet louver of the second tube row. Positive and negative values indicate rotation in opposite directions. For the small fin pitch \( F_p = 1.20 \) mm the X vorticity is much lower than for the large fin pitch \( F_p = 1.99 \) mm. The velocity vectors in the same plane are shown in Fig. 8. The small fin pitch clearly suppresses the vortex development. Thus for large fin pitches the generated vortices have a stronger enhancement effect compared to geometries with small fin pitches. This results in higher convective heat transfer coefficients and thus higher Colburn \( j \)-factors. Next to the reduced impact of the artificially created vortices by the delta winglets, the naturally occurring horseshoe vortices are also suppressed for the smallest fin pitch. This was observed during visualization experiments in a water tunnel [49]. The absence of horseshoe vortices also lowers the average Colburn \( j \)-factor. Thus the high Colburn \( j \)-factors corresponding to the smallest fin pitch \( F_p = 1.20 \) mm in Fig. 6a (for \( V_{in} > 1.2 \) m/s) are caused by the enhancement effect of the louver. The effect of the delta winglets is negligible. When increasing the fin pitch, the flow development length gets longer and thus the overall Colburn \( j \)-factor increases. The enhancement effect of the delta winglets also increases, while the enhancement effect caused by the louver reduces (the flow becomes more duct directed). As long as the effect of the louver geometry is dominant over the effect of the delta winglet geometry and the flow development, the Colburn \( j \)-factor decreases with increasing fin pitch at a fixed inlet velocity. From a certain fin pitch onwards, the distance between the fins is so large that the delta winglet effect outweighs the louver effect. When further increasing the fin pitch, the Colburn \( j \)-factor starts to increase again. This explains why the Colburn \( j \)-factor in Fig. 6a drops significantly when increasing the fin pitch from 1.20 to 1.71 mm, but then increases again when the fin pitch is further increased to \( F_p = 1.99 \) mm.

Fig. 6b shows that for \( V_{in} < 2.8 \) m/s, the highest friction factors correspond to the smallest fin pitch \( (F_p = 1.20 \) mm). In this case the fin density is high and the flow is more louver directed (high flow efficiency). For \( V_{in} > 2.8 \) m/s, however, the highest friction factor correspond with the largest fin pitch \( (F_p = 1.99 \) mm). The generation of longitudinal vortices thus causes additional pressure drop. For high velocities the vortices are stronger. Then the friction factors are higher than when no vortices are present (i.e. case 1 where the vortex development is suppressed). Huisseune et al. [25] explained that the form drag associated with the tube surface is reduced when vortices are generated because the separation point on the tube wall moves downstream. Fig. 6b illustrates that this reduction in form drag is smaller than the increase in frictional pressure drop and flow blockage caused by the delta winglet geometry: the net pressure drop increases when longitudinal vortices are generated.

A commonly used criterion to evaluate the thermal hydraulic performance of a heat exchanger is the area goodness factor. It is defined as the ratio of the Colburn \( j \)-factor to the friction factor. High values of \( j/f \) are preferred as this means less frontal area for
a fixed heat transfer and pressure drop [50]. The area goodness factor $j/f$ is plotted in Fig. 6c. Case 1 ($F_p = 1.20$ mm) should not be used at low velocities. The area goodness factor shows a very strong drop-off because the heat exchanger is oversized. In the low velocity range, the reference case ($F_p = 1.71$ mm) requires the smallest frontal area: the Colburn $j$-factor is smaller than for case 2 ($F_p = 1.99$ mm), but also the friction factor is significantly smaller. At higher velocities case 1 ($F_p = 1.20$ mm) is preferred.

LaHaye et al. [51] suggested evaluating the thermal hydraulic performance of heat exchangers by plotting the heat transfer performance factor $J$ (Eq. (9)) as function of the pumping power factor $F$ (Eq. (10)). They showed that for the same hydraulic diameter $D_h$, $J$ is proportional to the heat transfer per unit volume and $F$ is proportional to the pumping power per unit volume. $J$ vs. $F$ is thus a modification of the volume goodness factor proposed by London and Ferguson [52]. However, due to the variation of the fin pitch, the hydraulic diameter is not fixed for the three simulations considered here. The performance parameters developed by LaHaye et al. [51] can thus not be used. Soland et al. [53] modified the performance evaluation method of LaHaye et al. [51]. The heat transfer performance parameter $J_n$ and the pumping power performance parameter $F_n$ are given by Eqs. (11) and (12), respectively. The contraction ratio $\sigma$ is added because Soland et al. [53] based the Reynolds number on the inlet velocity instead of the maximum velocity in the heat exchanger. $J_n$ and $F_n$ are very similar to the performance parameters of LaHaye et al. [51]. However, the assumption of a fixed hydraulic diameter is eliminated. $J_n$ is plotted as function of $F_n$ in Fig. 6d. This allows a volume goodness comparison, which means a comparison in terms of the total heat transfer surface or core volume. The difference between the two highest fin pitches is small. Case 1 ($F_p = 1.20$) clearly shows a better volume goodness performance. Thus with a high fin density (i.e. small fin pitch) the heat exchanger can be made more compact for the same heat duty and pumping power.

$$J = j \cdot Re_{D_h}$$ (9)
$$F = f \cdot Re_{D_h}$$ (10)
$$J_n = \sigma \frac{j \cdot Re_{D_h}}{D_h^2}$$ (11)
$$F_n = \frac{f \cdot Re_{D_h}^3}{D_h^4}$$ (12)

In Fig. 6 the Colburn $j$-factors, friction factors and area goodness factors are plotted as function of the inlet velocity $V_{in}$. More commonly, these factors are plotted as function of the Reynolds number $Re_{D_h}$. The hydraulic diameter $D_h$ (Eq. (1)), however, is dependent on the fin pitch $F_p$. An increase in Reynolds number $Re_{D_h}$ can thus correspond with an increase in velocity, an increase in hydraulic diameter (and thus fin pitch) or a change of both. Because here the influence of the fin pitch was studied, it was preferred to present plots as function of the inlet velocity. The performance can easily be evaluated from these plots at a fixed velocity. For completeness, Fig. 9 shows the Colburn $j$-factor, friction factor and area goodness factor as function of the Reynolds number $Re_{D_h}$. At a fixed velocity, the Reynolds numbers of the reference case and case 1 are smaller.
compared to case 2 due to the smaller hydraulic diameter. Consequently, the Colburn j-curves are shifted towards smaller Reynolds numbers. In the rest of this paper all performance evaluation criteria are plotted as function of the Reynolds number $Re_Dh$, because for all the cases studied the hydraulic diameter is fixed (the hydraulic diameter is independent of the geometrical parameter under consideration).

6.2. Variation of the louver angle $\theta$

The effect of the louver angle on the Colburn j-factor and friction factor is plotted in Fig. 10a and b, respectively. The three louver angles considered are $\theta = 22^\circ$, $28^\circ$ and $35^\circ$. The highest Colburn j-factors are obtained for the largest louver angle (and thus high flow efficiency). When the louver angle is reduced from $35^\circ$ to $28^\circ$, the flow efficiency decreases. The flow is less aligned with the louvers and thus the Colburn j-factor and friction factor decrease. Huisseune et al. [54] also showed that less flow deflection results in weaker horseshoe vortices in the downstream tube rows. This also causes a lower Colburn j-factor and friction factor. When further reducing the louver angle from $28^\circ$ to $22^\circ$, the Colburn j-factor again increases. This is due to the presence of the delta winglets. For small louver angles the flow is strongly duct directed. In this case the $Y$ velocity component of the flow is small and the pressure difference across the delta winglet is larger than when the flow is strongly deflected (important $Y$ velocity component due to the large louver angle). Hence, for small louver angles the generated longitudinal vortices are stronger than for large louver angles. Furthermore, for a more duct directed flow the generated vortices also last further downstream as they are less affected by the deflected flow in the downstream tube row. As is shown in Fig. 10b, the increase in pressure drop associated with this heat

Fig. 11. Influence of the delta winglet angle of attack $\alpha$ on (a) the Colburn $j$-factor, (b) the friction factor, (c) the area goodness factor and (d) the modified volume goodness factor.

Fig. 12. Spanwise variation of the vorticity magnitude halfway the fin passage at a distance of $3/4F_p$ downstream of a tube in the second tube row ($V_{in} = 2.69$ m/s, $Re_{Dh} = 616$).

![Graphs showing variations of Colburn j-factor, friction factor, area goodness factor, and modified volume goodness factor with Reynolds number and louver angle.](image-url)
transfer enhancement is very small. As a result, the area goodness factor for $h = 2.2/C_{176}$ is high and thus the required frontal area is small. This is shown in Fig. 10c. In Fig. 10d the volume goodness comparison is presented. As the hydraulic diameters of the three geometries are identical, the performance evaluation method of LaHaye et al. [51] is used (Eqs. (9) and (10)). The smallest core volume corresponds with the largest louver angle ($h = 3.5/C_{176}$) where the enhancement is dominated by the louvers. The performance for $h = 2.2/C_{176}$ is only slightly smaller.

6.3. Variation of the delta winglet angle of attack $\alpha$

The effect of the delta winglet angle of attack on the Colburn $j$-factor and friction factor is plotted in Fig. 11a and b, respectively. The three angles of attack considered are $\alpha = 25^\circ$, $30^\circ$ and $35^\circ$. In plain fin heat exchangers with vortex generators, a larger angle of attack results in higher convective heat transfer coefficients because the vortices are stronger and thus the boundary layer is thinned more effectively and there is a better flow mixing [55]. In a compound design, however, the effect of the angle of attack on the thermal performance is different. Fig. 11a indicates that the highest Colburn $j$-factors occur for $\alpha = 30^\circ$. A smaller angle of attack ($\alpha = 25^\circ$) results in lower Colburn $j$-factors because the generated vortices are weaker. However, for a higher angle of attack ($\alpha = 35^\circ$), the Colburn $j$-factors are also lower compared to those for $\alpha = 30^\circ$. Fig. 12 shows the vorticity magnitude halfway the fin passage at a distance of $3/4F_p$ downstream of the tube in the second tube row as function of the spanwise length coordinate $z$ ($V_{in} = 2.69$ m/s, $Re_Dh = 616$). $z = 0$ corresponds to the center of the tube wake and $z \approx 2.0$ mm corresponds to the delta winglet position. In the tube wake ($z \approx 0$–1.0 mm) the vorticity magnitude is low. For higher $z$ the vorticity increases due to the generated vortices. An increase of the angle of attack $\alpha$ results in higher vorticity peaks and thus stronger vortices. But for larger $\alpha$ the vorticity peaks also shift away from the tube wake center. As the vortices are located further away from the tube wake center, their effect on the wake zone is smaller. This explains why for $\alpha = 35^\circ$ the Colburn $j$-factors are smaller than for $\alpha = 30^\circ$, even though the generated vortices are stronger.

It was expected that the pressure drop penalty is minimal for the smallest angle of attack as this results in smallest flow blockage. However, Fig. 11b shows that this is not the case. The smallest friction factors correspond to an angle of attack $\alpha = 35^\circ$. This again illustrates that the delta winglet vortex generators not only cause an increase in frictional pressure drop, but also reduce the form drag (as was explained by Huisseune et al. [25]). The optimal values of the delta winglet geometry correspond with the lowest overall pressure drop, including frictional pressure drop and form drag.

The area goodness factor and volume goodness factor are plotted in Fig. 11c and d, respectively. The smallest frontal area is required for $\alpha = 35^\circ$. The difference between $\alpha = 25^\circ$ and $\alpha = 30^\circ$ is small. The smallest heat exchanger volume is required for $\alpha = 30^\circ$. Notice that the influence of a variation of the angle of

Fig. 13. Influence of the delta winglet height ratio $h^*$ on (a) the Colburn $j$-factor, (b) the friction factor, (c) the area goodness factor and (d) the modified volume goodness factor.
attack reduces with increasing Reynolds number. This is consistent with the finding of Huisseune [48] that the delta winglets mainly contribute to the performance at low velocities.

6.4. Variation of the delta winglet height ratio $h^*$

Two delta winglet height ratios are considered: $h^* = 0.5$ and $h^* = 0.9$. The results are plotted in Fig. 13. The effect of the height ratio is very small, especially at high Reynolds numbers. This again illustrates that at high Reynolds numbers the thermohydraulics of the compound design are not dominated by the delta winglets [48]. From a thermal point of view the height ratio $h^* = 0.5$ seems preferential. Higher Colburn $j$-factors are penalized by increased friction factors. The reference case ($h^* = 0.9$) requires the smallest frontal area (highest area goodness factor). This is mainly due to the smaller friction factor. The difference in volume goodness is very small.

6.5. Variation of the delta winglet aspect ratio $A$

The effect of the delta winglet aspect ratio is plotted in Fig. 14. Two aspect ratios are considered: $A = 1.5$ and $A = 2.0$. The smallest aspect ratio results in the highest Colburn $j$-factor. Similar to the effect of the angle of attack $\alpha$, a too large aspect ratio results in vortices too far from the tube wake. Consequently, their enhancement effect on the wake zone is smaller. High Colburn $j$-factors are penalized by high friction factors. Taking the Colburn $j$-characteristic as well as the friction characteristic into account, the reference case ($A = 2.0$) requires the smallest frontal area, while case 8 ($A = 1.5$) can be made smaller in volume. Similar to the delta winglet angle of attack and height ratio, the influence of a variation of the aspect ratio on the thermohydraulics decreases with the Reynolds number.

As explained, the aspect ratio of the delta winglets may not be too large. On the other hand, the aspect ratio may also not be too small because then the delta winglets are completely located in

---

Table 3
Geometry used to study the influence of the spanwise position of the delta winglets.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Symbol</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Outer tube diameter</td>
<td>$D_o$ (mm)</td>
<td>6.75</td>
</tr>
<tr>
<td>Fin thickness</td>
<td>$t_f$ (mm)</td>
<td>0.12</td>
</tr>
<tr>
<td>Louver pitch</td>
<td>$L_p$ (mm)</td>
<td>1.50</td>
</tr>
<tr>
<td>Louver angle</td>
<td>$\alpha$ (°)</td>
<td>22</td>
</tr>
<tr>
<td>Fin pitch</td>
<td>$F_p$ (mm)</td>
<td>1.99</td>
</tr>
<tr>
<td>Transversal tube pitch</td>
<td>$T_p$ (mm)</td>
<td>1.76</td>
</tr>
<tr>
<td>Longitudinal tube pitch</td>
<td>$L_p$ (mm)</td>
<td>1.36</td>
</tr>
<tr>
<td>Streamwise DW position</td>
<td>$\Delta x$ (mm)</td>
<td>0.50, 0.30D, 0.40D</td>
</tr>
<tr>
<td>Spanwise DW position</td>
<td>$\Delta z$ (mm)</td>
<td>0.3D, 0.4D</td>
</tr>
<tr>
<td>DW angle of attack</td>
<td>$\alpha$ (°)</td>
<td>30</td>
</tr>
<tr>
<td>DW height ratio</td>
<td>$h^*$</td>
<td>0.7</td>
</tr>
<tr>
<td>DW aspect ratio</td>
<td>$A$</td>
<td>1</td>
</tr>
</tbody>
</table>
the wake zone and their enhancement effect is also smaller. However, for small aspect ratios the spanwise location of the delta winglet can be changed. This is discussed next.

6.6. Variation of the delta winglet position

In all previous simulations the delta winglets were placed in a common-flow-down orientation at a distance of $\Delta x = 0.5D_o$ and $\Delta z = \pm 0.3D_o$ from the tube center (see Fig. 3b). For plain fin heat exchangers with delta winglets, Pesteei et al. [16] found the best heat transfer to pressure drop performance for $\Delta x = 0.5D_o$ and $\Delta z = \pm 0.5D_o$. In compound designs $\Delta z$ has to be smaller than $0.5D_o$ due to the presence of the louvers. For an angle of attack $\alpha = 35^\circ/C176$ and an aspect ratio $K = 2$, the maximum $\Delta z$ possible is $0.3D_o$. However, $\Delta z$ can be larger if the angle of attack and aspect ratio are smaller. Two simulations were performed for the geometry listed in Table 3. In this case $\alpha = 30^\circ$ and $K = 1$. The spanwise positions of the delta winglets are $\Delta z = \pm 0.3D_o$ and $\Delta z = \pm 0.4D_o$. The streamwise position is fixed at $\Delta x = 0.5D_o$. The results are plotted in Fig. 15. An increase in $\Delta z$ results in an improved thermal performance, mainly at higher Reynolds numbers. Also the friction factor increases. Fig. 16a shows that for $\Delta z = \pm 0.3D_o$, all delta winglets are completely located in the wake zone. Then no longitudinal vortices are generated and thus the delta winglets do not cause any heat transfer enhancement. If the spanwise position of the delta winglets is increased to $\Delta z = \pm 0.4D_o$, the delta winglets of the first and second tube row are no longer located in the tube wakes (see Fig. 16b). The generated vortices reduce the wake zones and thus the local heat transfer is increased. In the third tube row, however, the delta winglets are still located in the tube wake. There the tube wake is wider than in the first two tube rows due to the absence of the effect of the staggered tube layout. Thus the delta winglets in the last tube row do not contribute to the heat transfer enhancement. This was also found in earlier work [25]. Considering the area goodness factor (Fig. 15c), $\Delta z = \pm 0.4D_o$ is preferred. For a given pumping power per unit volume, the heat transfer per unit volume is larger for $\Delta z = \pm 0.4D_o$ (Fig. 15d). Thus a smaller heat exchanger can be used. In general a larger $\Delta z$ is better from a thermal hydraulic point of view. In general, the gain is very small for the studied geometry because the delta winglets are small. The small aspect ratio was chosen to clearly illustrate the influence on the flow field when the vortex generators are completely located in the tube wakes: comparing Fig. 16a and b shows that the location of the delta winglets has a significant impact on the tube wake reduction.

The discussion above shows that it is important that the delta winglets are not completely located in the tube wakes. Different parameters have an effect here: the fin pitch (as this parameter affects the wake width), the delta winglet base, the spanwise winglet position $\Delta z$ and the delta winglet angle of attack $\alpha$. But also other parameters, which were not investigated in this work, might have an influence: the tube pitches (these also affect the wake dimensions, especially in a staggered tube layout), tube arrangement, number of louvers per tube row, etc. A more detailed optimization is necessary which takes more geometrical parameters into account and also the interactions between the different parameters. This is the subject of future work.

Fig. 15. Influence of the delta winglet position $\Delta z$ on (a) the Colburn $j$-factor, (b) the friction factor, (c) the area goodness factor and (d) the modified volume goodness factor.
Three-dimensional CFD simulations of a round tube heat exchanger with louvered fins and delta winglets were performed. The influence of five geometrical parameters on the thermal and hydraulic performance was investigated. The effect of the louver and delta winglet geometry results in conflicting design requirements. A small fin pitch and large louver angle cause a strong flow deflection and thus a large contribution of the louvers. But in this case the generation of longitudinal vortices is suppressed and thus the effect of the delta winglets is small. Also the delta winglet geometry itself has an important influence. A well-considered geometry and location of the delta winglets is essential for an improved performance. Care should be taken that the delta winglets are not completely located in the tube wakes, because in this case they do not cause any enhancement effect.

Acknowledgments

The authors would like to express gratitude to the FWO (Flemish Fund for Scientific Research) for the financial support.

References


Fig. 16. Velocity contours in a plane 0.15 s above the fin surface (V_{w} = 1.26 m/s).


