Abstract

A few decades ago, the product development process was just based on a trial and error procedure, and the designer’s experience. The need for a new way to design and manufacture more economical and sustainable products corroborates increasingly to a new vision of how to create new products for the benefit of society. Modern numerical tools allow greater knowledge about the physical phenomena involved in engineering problems and enable cost reduction with trials and time of manufacture and projection.

Among the equipment that can be mentioned where numerical simulation is used, can be found heat exchangers, which are capable of accomplishing the heat transfer between two fluid medias with different temperatures. Within the range of existing exchangers, this work will address a compact model with louvered fins, widely used in the automotive and aerospace industries, mainly due to their high thermal exchange surface vs occupied volume ratio. The heat exchanger surface is analised using computational fluid dynamics techniques disposable in the commercial code ANSYS CFX14® to reproduce the flow at service condition. Genetic optimization routines are used to increase the performance of heat exchanger. As a result, a heat transfer surface is obtained with about a 25% better performance according to the selected objective function. The dimensionless factor of the convective heat transfer coefficient (Colburn factor, $j$) and the friction factor (Fanning factor, $f$) used in (Wang et al., 1998), are employed for simulation. Experimental data are also used for validation.

**keywords:** compact heat exchanger, CFD, optimization, louver fins.

1. Introduction

The design of more efficient thermal equipment with lower cost is one of the goals of modern engineering. Processes and very sophisticated tools are used to increasingly achieve optimized and competitive components.

Thus, the use of numerical simulation and optimization algorithms become tools to achieve results, change the initial prototypes by virtual tests, reducing costs associated with the reduction of experimental trials (Herckert et al., 2004). The numerical analyses also allow better understanding of the phenomena involved.

In the present work a compact heat exchanger with louvered fins is analyzed by reproducing the flow at service condition using computational tools. These fins are fabricated by the stamping of sheet metal and the bending of the cut region, allowing a massflow between layers and increasing the heat transfer rate due to their influence on the flow and temperature behavior. Due to the effects caused by the louvered fins, it is necessary to perform CFD simulations as well as to apply optimization procedures to find the best arrangements of louvered fin angles.

Compact heat exchangers are frequently used in automotive and aerospace industries due to their compactness represented by a relationship between the heat transfer area and a volume higher than 700 m²/m³. The presented procedure is valid for incompressible flows and is restricted to the applied boundary conditions and geometrical restriction. However the procedure can be used in more refined studies considering other flow conditions and a full heat exchanger surface.
2. Materials and methods

The ANSYS CFX® commercial software is used to perform the fluid dynamic analysis with the finite volume method (FVM), based on the solution of equations of mass, energy and momentum conservation (Maliski, 2004), exhibited by equations 1, 2 and 3 respectively.

\[
\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_j} (\rho u_j) = 0 \tag{1}
\]

\[
\frac{\partial}{\partial t} (\rho u_j) + \frac{\partial}{\partial x_j} (\rho u_j u_i) = -\frac{\partial p}{\partial x_j} + \frac{\partial}{\partial x_j} \left( \mu \frac{\partial u_j}{\partial x_j} \right) + f_j + S_j \tag{2}
\]

\[
\frac{\partial}{\partial t} (\rho T) + \frac{\partial}{\partial x_j} (\rho u_j T) = \frac{\partial}{\partial x_j} \left( \frac{k}{\rho c_p} \frac{\partial T}{\partial x_j} \right) + S_T \tag{3}
\]

After the development of the computational procedure, the flow and temperature field variables are analyzed and the required dimensionless variables of interest (friction, j and Colburn, j factors) are also computed.

For turbulence modeling, the SST (Shear Stress Transport) turbulence model is used, which belongs to the family of RANS (Reynolds Averaged Navier-Stokes).

Developed by (Menter, 1994), this model was largely applied to calculate aerodynamic complex flows with adversely pressure gradients, as generally detected in airfoils.

Traditional models fail to capture these phenomena due to the degree of complexity and nonlinearity. To overcome these difficulties, the SST model (Menter, 1994), used two models, the k - ε and k - ω.

The k - ω model is applied to estimate the fluid characterization in regions near the wall where the flows are more complex, and the k - ε model is applied in regions far from the walls, where the turbulence phenomena are weaker and the shear stresses are lower, mainly because the ω property is sensible for these regions, reducing the precision of the model.

So, as the distance decreases in relation to the wall ε calculation, it is replaced by ω computation. The blending function is applied to command the alteration of these variables.

To use the SST model properly, it is necessary to adopt some quality criteria. Among them, the dimensionless distance parameter to the wall, y*, calculated by the following expression, must remain less than 1 in the mesh nodes near the wall for proper operation of the SST model. In equation 4, y is the distance from the wall, u* is the fluid friction velocity near the wall, v is the kinematic viscosity of the fluid and y' is the dimensionless distance.

\[
y^* = \frac{u^* y}{v} \tag{4}
\]
fluid. The behavior of the fluid flowing through the external cold side is one of the objects of the study performed in this work.

Compact heat exchangers are generally formed by a repeating geometric pattern. For this reason, the studied volume can be divided into reduced cells, and the analysis can be performed for a single cell device (Michael, 2006). This allows reduction of the computational effort and time spent waiting for the results. Consequently, optimization processes such as those utilized in this work that perform various simulations in series become feasible.

Used in this work are data obtained by (Wang et al., 1998), through physical analysis in specific equipment, and the numerical simulations performed by (Jang and Chen, 2013), in commercial code as presented herein.

For domain creation, sections are inserted before the inlet of cell (one time the internal diameter) and after the end (seven times the internal diameter) to prevent abrupt behavior variations, as recommended by (Perrotin and Clodic, 2004), to facilitate the convergence.

The main dimensions utilized for heat exchanger domain are shown in Figure 1 and its values are presented in Table 1.

The element type utilized is the tetrahedral element, recommended for its adaptability to complex geometries. To reach a value of $y’$ near 1, the edges of elements are computerized as 0.5 millimeter, resulting in a mesh around 1.97E+5 elements and 6E+4 nodes. Details of mesh are illustrated in Figures 2 and 3.

The Reynolds number is calculated by equation 5 and it is used as an output variable, to compute the dimensionless numbers $f$ and $j$.

$$Re = \frac{\rho U_{\text{max}} L}{\mu}$$  \hspace{1cm} (5)

- $U_{\text{max}}$: means the maximum velocity in vertical section with smaller area – the flow is considered incompressible due to the low velocity values.
- $L$: is the characteristic length

In the entrance domain, the air flows from 0.50 to 7.50 m/s with 300,0K. The tubes have a wall temperature of 353.0K.

The boundary conditions are displayed in Figures 4 and 5. At the wall surfaces of tubes and fins, the no slip boundary condition is applied. This condition assumes that the fluid velocity at the wall equals to zero. Symmetry is assumed in the lateral faces of the domain and in the upper and bottom faces of the domain, a periodic condition is used.
In the post-processing phase, after solving the systems of equations, the output variable generated is the Reynolds number, obtained by equation 2 and the friction and Colburn factors $f$ and $j$, obtained by a procedure developed by (Wang et al., 1998). These authors experimentally analyzed 49 louvered-fin heat exchangers and developed the $f$ and $j$ correlations used in this work. The reference (Weber, 2007), found similar correlations for Colburn and fanning factors by physical tests.

For a low Reynolds number (lower than 1000), the $j_{low}$ is calculated by equation 6 using the exponents of equations 7, 8, 9 and 10 ($J_1$, $J_2$, $J_3$ and $J_4$).

\[ j_{low} = 14,3117 \left( \frac{Re}{D_c} \right) J_1 \left( \left( \frac{F_p}{L_p} \right) J_2 \left( \frac{F_h}{P_t} \right) J_3 \left( \frac{P_t}{P_l} \right) \right)^{1,724} \]  

(6)

\[ j_1 = -0,991 + 0,1055 \left( \ln \left( \frac{L_h}{L_p} \right) \right) \left( \frac{P_p}{P_t} \right)^{3,1} \]  

(7)

\[ j_2 = -0,7344 + 2,1059 \left( \ln \left( \frac{L_h}{L_p} \right) \right) \left( \ln \left( \frac{Re}{D_c} \right) - 3,2 \right) \]  

(8)

\[ j_3 = 0,08485 \left( \frac{P_p}{P_t} \right)^{4,4} N^{0,68} \]  

(9)

\[ j_4 = -0,1741 \ln (N) \]  

(10)

For high Reynolds number (higher than 1000), the $j_{high}$ is calculated by equation 11 using the exponents of equations 12, 13, 14 and 15 ($J_9$, $J_{10}$, $J_{11}$ and $J_{12}$).

\[ j_{high} = 1,1373 \left( \frac{Re}{D_c} \right) J_9 \left( \left( \frac{P_p}{L_p} \right) J_{10} \left( \frac{L_h}{L_p} \right) J_{11} \left( \frac{P_t}{P_l} \right) N^{0,3545} \right) \]  

(11)

\[ j_9 = -0,6027 + 0,02593 \left( \frac{P_p}{D_h} \right)^{0,52} \left( \ln \left( \frac{L_h}{L_p} \right) \right) N^{-0,5} \]  

(12)

\[ j_9 = -0,4776 + 0,40774 \left( \ln \left( \frac{Re}{D_c} \right) - 4,4 \right) \]  

(13)
(14) 
\[ j = -0.58655 \left( \frac{F_p}{D_h} \right)^{2.3} \left( \frac{P_l}{P_t} \right)^{N - 0.65} \]

(15) 
\[ j = 0.0814 \left( \ln (Re) \right) - 3 \]

In the same way, the \( f \) factor is calculated for the low and high Reynolds number. For \( Re \) lower 1000, \( f_{low} \) is calculated by equation 16 with exponents \( F_1, F_2, F_3, \) and \( F_4 \) extracted from equations 17 to 20.

(16) 
\[ f_{low} = 0.00317 \left( \frac{F_p}{P_t} \right)^{F_1} \left( \frac{D_h}{D_l} \right)^{F_2} \left( \frac{L_h}{L_p} \right)^{F_3} \left( \ln \left( \frac{A_o}{A_t} \right) \right)^{F_4} \cdot 6.0483 \]

(17) 
\[ F_1 = 0.1692 + 4.4118 \left( \frac{F_p}{P_t} \right)^{0.3} \left( \frac{L_h}{L_p} \right)^{2} \left( \ln \left( \frac{P_l}{P_t} \right) \right) \left( \frac{F_p}{P_t} \right)^{3} \]

(18) 
\[ F_2 = -2.6642 - 14.3809 \left( \frac{1}{\ln (Re)} \right) \]

(19) 
\[ F_3 = -0.6816 \left( \ln \left( \frac{F_p}{P_t} \right) \right) \]

(20) 
\[ F_4 = 6.4668 \left( \frac{F_p}{P_t} \right)^{1.7} \left( \ln \left( \frac{A_o}{A_t} \right) \right) \]

\( f_{high} \) is found by equation 21 with exponents \( (F_{19}, F_{20}, F_{21}, F_{22}, \) and \( F_{23} \)) of equation 22 to 26.

(21) 
\[ f_{high} = 0.06393 \left( \frac{F_p}{P_t} \right)^{F_5} \left( \frac{D_h}{D_l} \right)^{F_6} \left( \frac{L_h}{L_p} \right)^{F_7} \left( \ln (Re) \right)^{F_8} \cdot 0.65 \]

(22) 
\[ F_5 = 0.1395 - 0.0101 \left( \frac{F_p}{P_t} \right)^{0.58} \left( \frac{L_h}{L_p} \right)^{2} \left( \ln \left( \frac{A_o}{A_t} \right) \right) \left( \frac{P_l}{P_t} \right)^{1.9} \]

(23) 
\[ F_6 = -6.4367 \left( \frac{1}{\ln (Re)} \right) \]

(24) 
\[ F_7 = 0.05875 \left( \ln (Re) \right) \]

(25) 
\[ F_8 = -2.0585 \left( \frac{F_p}{P_t} \right)^{1.67} \left( \ln (Re) \right) \]

(26) 
\[ F_9 = 0.1036 \left( \ln \left( \frac{P_l}{P_t} \right) \right) \]

The values of the output variables \( f \) and \( j \) are compared with the experimental data obtained in (Wang et al., 1998). After obtaining accurate results for these variables, optimization of heat exchanger geometry is performed. The design variable (DV) considered is the inclination angle of louver fins of domain, as used by (Stephan, 2002; Ameel et al., 2012).

To execute the optimization study, the commercial code DesignExplorer from Ansys is used, selecting the the MOGA procedure. This is an algorithm of evolutionary type, that uses natural selection to generate and choose geometry more efficiently, according to the established objective. There are seven DVs employed which are the fins angles, shown in figure 6.

Figure 6
Design variables used for optimization.
To generate the first generation of individuals, the design of experiments of type Optimal Space Filling was used, creating 83 different geometries. The subsequent generations are created selecting the best individuals of previous populations, chosen by maximization of the objective function described below in equation 27.

This expression allows recognizing the global alterations that occurs within the domain, avoiding the use of locally defined factors. The temperature difference in the numerator is used because its augmentation means that more energy was transferred to the cold chain. For the denominator, the pressure differential is used because there is an interest in reducing the pressure losses. So the $G$ expression formed by these two quantities will be maximized by the optimizer algorithm.

$$ G = \frac{\Delta T}{\Delta p} $$

(27)

3. Results of analysis

Through the expressions 3 to 23 and the Reynolds number obtained in simulation, the Colburn and Fanning factors are calculated and displayed in Figure 7. The curves of numerical results are exhibited with the values obtained from correlations by (Wang et al., 1998).

In Figure 7, the red line represents the experimental data, the black lines represent the numerical values and the green lines represent the experimental error variation of 15% from the experimental test. The simulation reached good accuracy in relation to physical model by (Wang et al., 1998).

Flow fields and temperature distribution are displayed in Figures 8, 9 and 10, respectively. As can be seen, the points with more intense turbulence are located in the vicinity of V-shaped fins.
To prove that the SST turbulence model is working correctly, Figure 11 shows the distribution of the mixing function values. When the mixing function reaches values below 1, the model $k$ - $\omega$ is executed along the wall and the model $k$ - $\omega$ in the rest domain. The blended function contour proves that $k$ - $\omega$ model is executed near the wall as expected.

In Figure 12 is shown the profile of the first generated population using the statistical tool specified in the text. Each broken line represents an individual and each vertical axis represents the value of design variable. The intersection of break line and the vertical axis exhibit a value of this variable for that geometry. The chaotic appearance of this graph demonstrated the variability of individuals in first generation. A good variability of characteristics increases the chances of finding betters performances for heat exchangers.

After the optimization step, the 3 best geometries in all population generated are found, with angles of each louver, exhibited in Table 2. The maximum performance reached was a 25% increase of the objective function. The analysis of sensibility of design variables in Figure 13, allowed to find the most important fin angles that influence the value of objective function. For this optimization routine, the fifth and seventh are the most influential for optimization results.
Application of optimization for improvement of the efficiency of louvered-fin compact heat exchangers

Figure 13
Analysis of DV sensibility.

<table>
<thead>
<tr>
<th>Angles of Lovers</th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
<th>6</th>
<th>7</th>
<th>$\Delta G$ [$^\circ$C/Pa]</th>
<th>% increase</th>
</tr>
</thead>
<tbody>
<tr>
<td>Individual A</td>
<td>50.1</td>
<td>50.0</td>
<td>50.0</td>
<td>50.2</td>
<td>59.9</td>
<td>40.9</td>
<td>40.8</td>
<td>0.0046</td>
<td>25.61</td>
</tr>
<tr>
<td>Individual B</td>
<td>50.0</td>
<td>50.0</td>
<td>50.0</td>
<td>50.4</td>
<td>58.4</td>
<td>40.6</td>
<td>40.1</td>
<td>0.00454</td>
<td>23.97</td>
</tr>
<tr>
<td>Individual C</td>
<td>50.0</td>
<td>50.3</td>
<td>50.2</td>
<td>51.9</td>
<td>59.7</td>
<td>42.8</td>
<td>43.5</td>
<td>0.00439</td>
<td>19.87</td>
</tr>
</tbody>
</table>

4. Conclusion

The methodology used in the present paper is efficient for the improvement of the simulated component.

The SST turbulence model provided good results when compared to the experimental data and the MOGA evolutionary algorithm provided an increase of approximately 25% for the objective function.

5. Acknowledgements

The authors fully acknowledge the support received from CNPq, FAPEMIG and UFSJ.

6. References


Received: 25 January 2016 - Accepted: 10 May 2016.