CFD ANALYSIS OF FLOW THROUGH BRAZED PLATE HEAT EXCHANGER

Oana GIURGIU, Angela PLESA, Lavinia SOCACIU

Abstract: The aim of this study is to determine if the geometry proposed for the channel suites in case of a brazed heat exchanger with high heat flow and low values for pressure losses. For this, in Solid Works software, were designed two plate geometries with different crimping angles, in order to form a channel. Using Computational Fluid Dynamics numeric simulation was analyzed the influence of the geometry on the fluid distribution and pressure drops in the proposed geometry of the channel. The analysis of the velocity and pressure drops fields were achieved with the aid of simulation software Fluent.

Key words: plate heat exchanger, velocity field, CFD analysis.

1. INTRODUCTION

Compact brazed plate heat exchangers (BPHEs) are widely used in refrigeration, air conditioning and industrial applications owning to features such as high efficiency, compactness, low volume to area ratio, low refrigerant charge [1].

Since the flow distribution is dependent of velocity, it is also dependent of Reynolds number. A higher velocity gives a higher Reynolds number. Flow is either laminar or turbulent depending on the flow conditions. Reynolds number has been developed as a criterion to determine whether flow in a studied domain is laminar or turbulent. It describes the relation between inertial forces, in terms of density ($\rho$), average velocity ($w$) and hydraulic diameter ($D_h$), to viscous forces in terms of dynamic viscosity ($\mu$). According to [2], internal flow is laminar if the Reynolds number is less than 2300, transitional between 2300 and 4000, and turbulent if higher than 4000.

Reynolds number can be calculated with:

$$Re = \frac{\rho \cdot w \cdot D_h}{\mu}$$  \hspace{1cm} (1)

where: \(\rho\) is the density, \(w\) is the average velocity, \(D_h\) is the hydraulic diameter, \(\mu\) is the dynamic viscosity.

The hydraulic diameter ($D_h$) for a channel of this shape is defined by:

$$D_h = \frac{4 \cdot A_{TC}}{P}$$  \hspace{1cm} (2)

where: $A_{TC}$ – heat transfer area and $P$ is the wetted perimeter.

For this shape, the Reynolds number can be calculated with:

$$Re = \frac{2 \cdot m}{\mu \cdot l}$$  \hspace{1cm} (3)

where: \(m\) represent the mass flow rate and \(l\) is the width of the plate.

The aim of this research paper was to simulate the fluid flow between a channel with a different geometry. Were used two types of plates, one with corrugated angle $\beta=30^\circ$ and another with $\beta=60^\circ$.
2. NUMERICAL ANALYSIS

2.1 Plate model

Using the Computational Fluid Dynamics (CFD) analysis was studied the fluid flow for a combined type of channel formed between two types of plates, one with corrugated angle $\beta=30^\circ$ and another with $\beta=60^\circ$.

Having in mind the purpose of this study to find the way in which the geometry of the plates is influencing the liquid distribution in the channel formed between the plates, we made some assumption:
- was considered that the cold fluid flows through the channel;
- plate has a constant temperature, that of the hot fluid.

The Re number was 3500 for the studied flows based on stream wise mean velocity and equivalent diameter of the flow cross section.

From previous studies, we concluded that the work fluid should be distributed evenly over the entire area studied. For this, were introduced two moderation areas (1 and 3 in Figure 2), one at the entrance and one at the exit of the plate [3].

![Fig. 2. Geometric model: 1 – inlet area, 2 – heat transfer area, 3 – exit area](image)

The analyzed geometric pattern was modelled in Solid Works, then imported in Ansys Fluent where was created the geometric mesh. Meshing of a 3D domain is usually done with tetrahedral, hexahedral, triangular prisms (wedges) quadrilateral pyramids [4] or irregular polyhedral.

2.2 Turbulence model

Turbulence model analysis led to the following conclusions on the advantages and disadvantages, as described in Table 1.

Because of the advantages shown in Table 1 the $k - \omega$ SST turbulence model was chose. Equations underlying the model of turbulence are the kinetic energy transport equation (relation 4) and the transport equation for turbulent dissipation rate (relation 5):

$$\frac{\partial k}{\partial t} + u \frac{\partial k}{\partial x} + \frac{\partial}{\partial y} \left( \frac{\partial k}{\partial y} \right) + w \frac{\partial k}{\partial z} = \Gamma_k \left( \frac{\partial^2 k}{\partial x^2} + \frac{\partial^2 k}{\partial y^2} + \frac{\partial^2 k}{\partial z^2} \right) + G_k - Y_k.$$

(4)

$$\frac{\partial \omega}{\partial t} + u \frac{\partial \omega}{\partial x} + \frac{\partial}{\partial y} \left( \frac{\partial \omega}{\partial y} \right) + w \frac{\partial \omega}{\partial z} = \Gamma_\omega \left( \frac{\partial^2 \omega}{\partial x^2} + \frac{\partial^2 \omega}{\partial y^2} + \frac{\partial^2 \omega}{\partial z^2} \right) + G_\omega - Y_\omega - D_\omega.$$

(5)

where: - $u$ – fluid velocity, in [m s$^{-1}$]; - $\rho$ – fluid density, in [kg m$^{-3}$]; - $p$ – fluid pressure, in [N m$^{-2}$]; - $\nu$ – cinematic viscosity, in [m$^2$ s$^{-1}$]; - $G_k$ - is the generation coefficient of turbulent kinetic energy due to gradients of medium velocity; - $G_\omega$ - represent the specific rate of dissipation for turbulent kinetic energy; - $\Gamma_k$ și $\Gamma_\omega$ represents the effective diffusion of $k$, respectively $\omega$; - $Y_k$ și $Y_\omega$ are the effective diffusion of $k$ and $\omega$; - $D_\omega$ - is a diffusion coefficient; - cartesian coordinate system is used.

Table 1

<table>
<thead>
<tr>
<th>The turbulence model</th>
<th>Advantage</th>
<th>Disadvantage</th>
</tr>
</thead>
<tbody>
<tr>
<td>$k - \varepsilon$</td>
<td>It presents a good accuracy is robust and does not require large memory resources of the computer.</td>
<td>Moderate results for complex flows.</td>
</tr>
<tr>
<td>$k - \omega$</td>
<td>Applicable for flow around obstacles and flows with boundary layer.</td>
<td>It's necessary a fine mesh grid of the boundary layer. In free flow areas, the results are mediocre.</td>
</tr>
<tr>
<td>$k - \omega$ SST</td>
<td>It combines two patterns of turbulence, so for the boundary layer areas it uses the $k - \omega$ model, and for the free flow areas the $k - \varepsilon$ model.</td>
<td>A fine meshing mesh grid is required. To process results requires large computer memory resources.</td>
</tr>
</tbody>
</table>
2.3 Mesh

Considering a three-dimensional flow of fluid through the channel of the heat exchanger plates shown in Figure 3 and a constant heat flux being applied at the plates we start the numerical modelling. We used SST $k-\omega$ turbulence model, because this turbulence model is well suited for simulating flow in the viscous sub-layer and to predict flow behavior in regions away from the wall [5].

In this study we used a "2nd order upwind" to solve the system of five equations. Regarding the accuracy of the results, it was imposed a criterion of convergence for all residual variables of $10^{-4}$ [6].

The mesh for analyzed channel (Figure 3) was performed using tetrahedral elements, thinners in the entry areas and dense in the wavy areas. To achieve the grid were used approximately 14 million tetrahedral elements.

The numerical simulation indicates that the central working fluid layer follows the length of the flow channel formed between the two plates over their entire active length.

The fluid layers near the plate walls follow the drawn profile of the channel by the inclination angle $\beta$, after which it intersects and mixes with the remainder of the fluid, favoring the formation of the turbines in the fluid mass.

3. RESULTS AND DISCUSSION

In figure 4 is presented the vector representation of the velocity profile, resulted from the CFD Analysis. The corrugated shape of the flow channel prints turbulence in the fluid flow, which is noticeable by the increased intensification of the velocity vectors in these areas. This confirms that after the liquid enters a corrugated area, most of it follows the flow path until it reaches the side wall, and is again directed to the corrugated area, a behavior similar to that resulting from the CFD simulation made.

This confirms that after the liquid enters a corrugated area, most of it follows the flow path until it reaches the side wall, and is again directed to the corrugated area, a behavior like that resulting from the CFD simulation made.

The pressure drop distribution in the analyzed channel shown in Figure 6 indicates that the highest recorded values are 0.36 bar, acceptable values for brazed heat exchangers.
4. CONCLUSION

This paper presents a numerical study CFD type over the plate of heat exchangers. For this study were designed two models of plates with 30 and 60 degrees value of corrugation angle, and after were assembly in order to form a fluid channel. The results obtained from CFD simulation indicate that this type of flow channel can be used in brazed heat exchangers.

5. REFERENCES