TECHNICAL UNIVERSITY OF CLUJ-NAPOCA

ACTA TECHNICA NAPOCENSIS

Series: Applied Mathematics, Mechanics, and Engineering
Vol. 61, Issue IV, November, 2018

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 (ρ), average velocity (w) and hydraulic diameter (D_h), to viscous forces in terms of dynamic viscosity (μ). 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.

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

Due to the complex geometries of the flow channels, Reynolds number in plate heat exchangers is often defined differently than for rectangular ducts. A cross section of a plate heat exchanger flow channel is represented in Figure 1. The colored area between the wavy shaped curves, represents the fluid domain with a main flow direction normal to the cut.

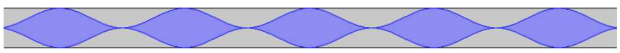

Fig. 1. Cross section of a plate heat exchanger

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

$$D_h = \frac{4 \cdot A_{TC}}{P} \quad (2)$$

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

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

$$Re = \frac{2 \cdot \dot{m}}{\mu \cdot l} \quad (3)$$

where: $\dot{m}$ 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 β=30° and another with β=60°.

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].

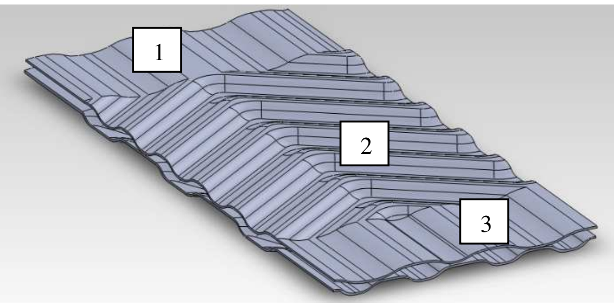

Fig. 2. Geometric model: 1 – inlet area, 2 – heat transfer area, 3 – exit area

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 – ω 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):

$$\frac{\partial k}{\partial t}+u\frac{-\partial k}{\partial x}+\vartheta\frac{-\partial k}{\partial y}+w\frac{-\partial k}{\partial z}= \\ =\Gamma_k\left(\frac{\partial^2 k}{\partial x^2}+\frac{\partial^2 k}{\partial y^2}+\frac{\partial^2 k}{\partial z^2}\right)+G_k-Y_k. \tag{4}$$

$$\frac{\partial \omega}{\partial t}+u\frac{-\partial \omega}{\partial x}+\vartheta\frac{-\partial \omega}{\partial y}+w\frac{-\partial \omega}{\partial z}= \\ =\Gamma_\omega\left(\frac{\partial^2 \omega}{\partial x^2}+\frac{\partial^2 \omega}{\partial y^2}+\frac{\partial^2 \omega}{\partial z^2}\right)+G_\omega-Y_\omega+D_\omega. \tag{5}$$

where: - u – fluid velocity, in [m s^{-1}]; - ρ – fluid density, in [kg m^{-3}]; - p – fluid pressure, in [N m^{-2}]; - υ – cinematic viscosity, in [m^2 s^{-1}]; - G_k - is the generation coefficient of turbulent kinetic energy due to gradients of medium velocity; - G_ω - represent the specific rate of dissipation for turbulent kinetic energy; - Γ_k şi Γ_ω represents the effective diffusion of k, respectively ω; - Y_k şi Y_ω are the effective diffusion of k and ω; - D_ω - is a diffusion coefficient; - cartesian coordinate system is used.

Table 1

Comparison of analysed turbulence models

The turbulence model	Advantage	Disadvantage
k – ε	It presents a good accuracy is robust and does not require large memory resources of the computer.	Moderate results for complex flows.
k – ω	Applicable for flow around obstacles and flows with boundary layer.	It's necessary a fine mesh grid of the boundary layer. In free flow areas, the results are mediocre.
k – ω SST	It combines two patterns of turbulence, so for the boundary layer areas it uses the k - ω model, and for the free flow areas the k - ε model.	A fine meshing mesh grid is required. To process results requires large computer memory resources.

	This model is suited to the study of plate heat exchangers for which the flow must be modelled in detail in both the central and wall areas.	

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 - ω 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 β, after which it intersects and mixes with the remainder of the fluid, favoring the formation of the turbines in the fluid mass.

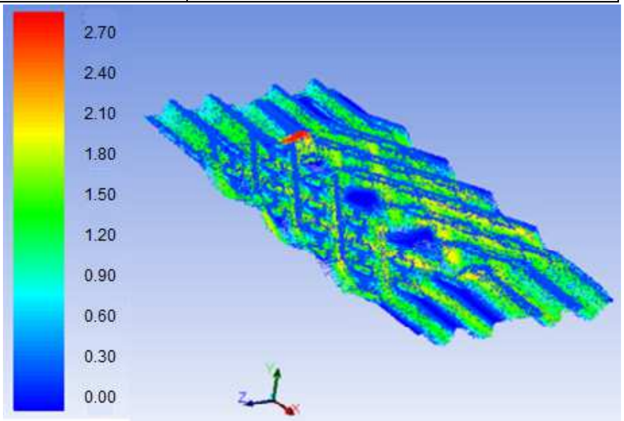

Fig. 4. Vector representation of the velocity profile

From the velocity field analysis, it can be noticed that the corrugated shape of the flow channel imparts turbulence to the fluid flow, which is visible through the increased intensification of velocity vectors in these areas (Figure 5).

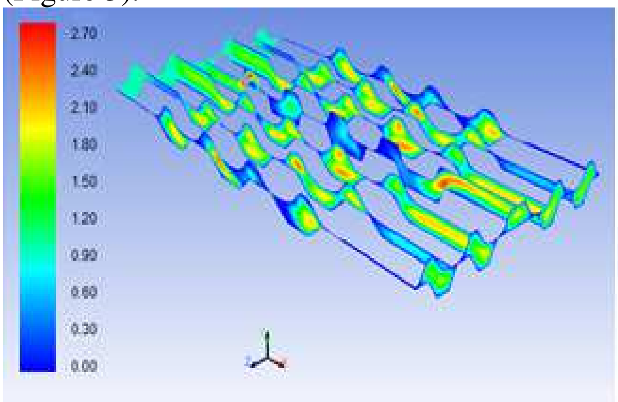

Fig. 5. Velocity field

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.

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.

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 like that resulting from the CFD simulation made.

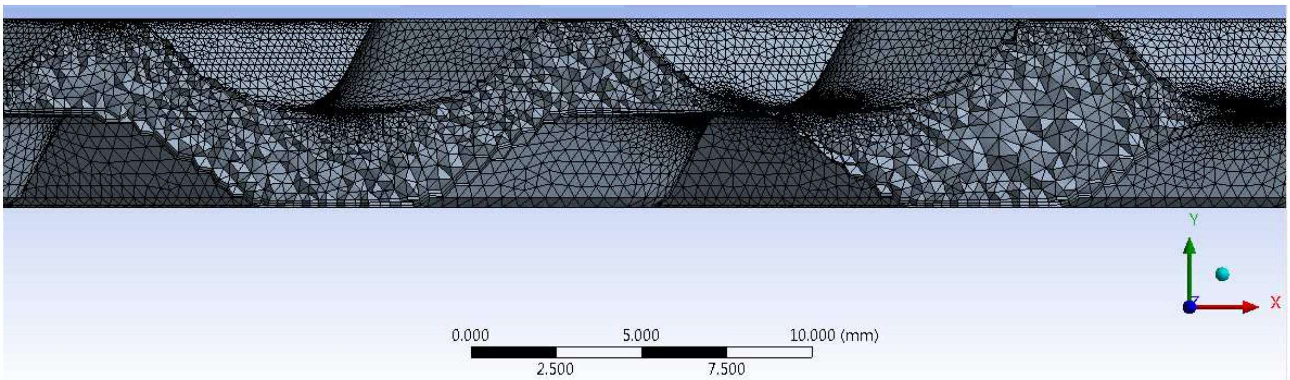

Fig. 3. Mesh detail

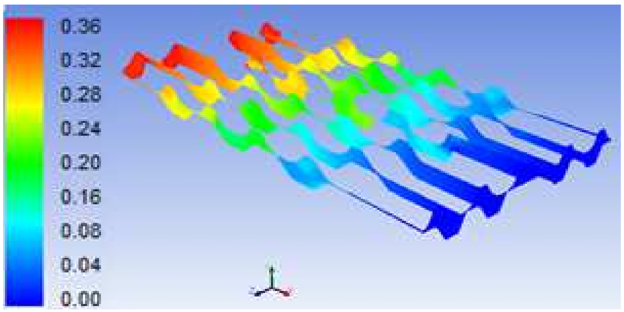

Fig. 6. Pressure field

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

[1] Gullapalli V. S., Mecke A., *Modeling of brazed plate heat exchangers for ORC systems*, Energy Procedia 129, pp. 443-450, 2017.

[2] Cengel Y., Cimbala J., *Fluid Mechanics - Fundamentals and Applications*. (2nd edn.). New York: McGraw-Hill, 2010.

[3] Giurgiu O., Pleşa A., Socaciu L., *Plate heat exchangers–flow analysis through mini channels*, Energy Procedia 85, pp. 244-251, 2016.

[4] Zhao Z.N., *Effects of the corrugated inclination angle on heat transfer and resistance performances of plate heat exchangers*, Petro-Chemical Equipment, 5, 2-3, 2001.

[5] Meslem A., Bode F., Croitoru C., Nastase I., *Comparison of turbulence models in simulating jet flow from a cross-shaped orifice*, European Journal of Mechanics B/Fluids, 2014.

[6] Salim M., Salim, Cheah S.C., *Engineers and Computer Scientists* 2, 2009.

Analiza CFD a curgerii prin schimbătoarele de căldura brazate

Rezumat: Scopul acestui studiu este de a determina dacă geometria propusă pentru canalul de curgere propus este potrivita în cazul unui schimbător de căldură brazat, cu o turbulenta ridicata şi valori scăzute pentru pierderile de presiune. Pentru aceasta, în software-ul Solid Works, au fost proiectate două geometrii de placă cu diferite unghiuri de inclinare, pentru a forma un canal de curgere. Folosind simularea numerică de tip Computational Fluid Dynamics, a fost analizată influența geometriei canalului de curgere asupra distribuției fluidului şi a căderilor de presiune. Analiza distribuției câmpurilor de viteza şi acelui de presiune a fost realizată cu ajutorul software-ului de simulare Fluent.

Oana GIURGIU, Ph.D. Eng., Assistant Teacher, Technical University of Cluj-Napoca, Mechanical Engineering Department, E-mail: oana.giurgiu@termo.utcluj.ro, No.103-105, Muncii Blvd., 400641, Cluj-Napoca

Angela PLESA, Ph.D. Eng., Teacher, Technical University of Cluj-Napoca, Mechanical Engineering Department, E-mail: angela.plesa@termo.utcluj.ro, No.103-105, Muncii Blvd., 400641, Cluj-Napoca

Lavinia SOCACIU, Ph.D. Eng, Lecturer, Technical University of Cluj-Napoca, Mechanical Engineering Department, E-mail: lavinia.socaciu@termo.utcluj.ro, No.103-105, Muncii Blvd., 400641, Cluj-Napoca