2016 International Conference on Power, Energy Engineering and Management (PEEM 2016)
ISBN: 978-1-60595-324-3

Advanced Methods of Modelling and Design of Plate Heat Exchangers

Vaclav DVORAK*

Technical University of Liberec, Studentska 2, 46117, Czech Republic

*Corresponding author

Keywords: Heat exchanger, CFD, Dynamic mesh.

Abstract. Research of devices for heat recovery is currently focused on increasing the heat effectiveness of plate heat exchangers. The goal of optimization is not only to increase the heat transfer but also reduce the pressure loss and material costs. During the optimization of plate heat exchangers using CFD, we are struggling with the problem of how to create a quality computational mesh inside complex and irregular channels. These channels are formed by combining individual plates or blades that are shaped by molding, vacuum forming, or similar technology. Creating computational mesh from the bottom up manually is time consuming and does not help later optimization. The paper presents a new method for creating of computational meshes to simulate flow and heat transfer in a plate heat exchanger. This method is based on dynamic mesh method "user defined deforming" provided by ANSYS Fluent. The method is fast and provide applicable tool for optimization. Influences of cell size and count of layers of computational cells on heat transfer and pressure drop were investigated. It was found that the minimum number of layers is twelve across the channel, otherwise obtained data can be irrelevant.

Introduction

Heat exchangers belong to the main parts of each energy and thermal devices and systems. The main purpose of each heat exchanger is to transport heat energy from one warmer medium, such as gas, liquid or steam, into other colder medium. In recuperative heat exchangers, both media are separated by walls and the heat energy must transfer also through this material.

The phenomena in heat exchanger are very complicated and include not only heat conduction in material, which separates both mediums, but also flow of both mediums and heat transfer from medium into material walls. The development of recuperative heat exchangers in recent years focused on increasing effectiveness, but others requirements must be fulfilled too. We will focus mainly on plate heat exchangers, in which both mediums are separated by plates, in this study.

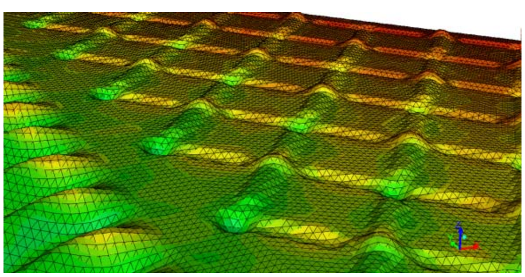

Figure 1. Example of an undulated heat exchange surface.

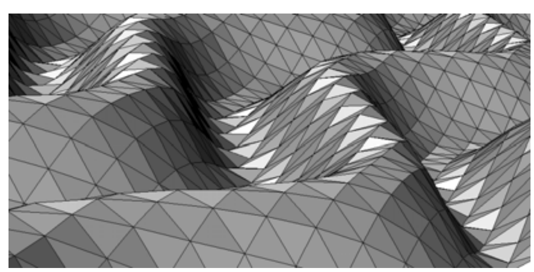

Figure 2. Rough surface with high curvature.

There are two basic characteristic properties typical for each heat exchanger – it is the effectiveness and pressure loss. To increase the efficiency of heat transfer, the heat exchange surface of the exchanger is provided with grooves or undulations. The purpose of such elements is to define the plate pitch and also to disturb both velocity and temperature boundary layers and thus intensify the heat transfer between the fluid and the heat transfer surface. An example of such undulation is in Figure 1. However, the slope of the undulation is limited by possibilities of shaping the used material and the intensification of heat transfer is limited by the maximum pressure loss that is available. As a result, we face a wide field for shape optimizing the heat exchange surface of each heat exchanger.

Computational fluid dynamic (CFD) methods and software are commonly used to calculate flow and heat transfer in plate heat exchangers, but this task is not trivial. Complicated and irregular narrow channels are created by assembling the heat exchanger from corrugated plates. Consequently, creating a computational mesh into this complicated channels is difficult task.

Therefore, most of researchers that dealt with design and investigation of performance of plate heat exchangers based on numerical calculations used the unstructured mesh. E.g. Gherasim et al. in work [1] presented the comparison of various grids for plate heat exchanger modelled by tetrahedral mesh. In order to assess the influence of the grid resolution on the solution, five grids were created and tested by meshing the volumes with different interval sizes. It was founded that the two grids with smallest elements give very close results. There are some next researchers who dealt with numerical simulations of plate heat exchangers with the chevron (undulated) profile. E.g. Tsai [2], Liu [3] dealt with these heat exchangers with different geometries. The conclusions about temperature and pressure drop were similar to Gherasim [1]. Novosád in work [4] investigated the influence of oblique waves on the heat transfer surface. About 80 different variants of undulation were calculated. Each option was modeled separately, meshed, loaded into the solver, set the boundary conditions and subsequently evaluated by calculation. Novosád et al. in work [5] also investigated influence of different mesh sizes and quality. Results were again similar to that made by Gherasim [1].

Disadvantages of repeated generation of computational meshes are: It is slow, it can takes several days to create proper 3D model of a plate and other days to create mesh on the model. Meshes made in different models are not similar, creating of appropriate mesh can be impossible and parameterization of the model is problematic. Further, even a small change of geometry requires to go through the whole process of model creation and mesh generation again. As a result, there is high probability of creation errors of model and low quality of mesh cells. Furthermore, meshes are not similar, i.e. the size, shape, and height of wall adjacent cells are not the same for different topologies which amplify numerical errors. Most of these disadvantages complicate significantly the optimization process or even disable it. Therefore, the aim of this work is to present a new method for creation of computational meshes and to show influence of mesh density on results of numerical calculation.

Methods

Methods of Grid Generation

The first step in our new way to create a model of a real complicated plate heat exchanger is creation a simple model prepared for CFD simulation. This model has the same outline and dimensions as a real heat exchanger, but the geometry of the plates is the simplest one – flat. We can create this model in common software like ANSYS Design modeler. In the simplest case, the model has only one plate with zero thickness, two inlets and two outlets, and two planes parallel to the plate which act as symmetrical boundary conditions. A triangle surface mesh has to be used, because each vertex will move independently and squares would collapse during this procedure, see Figure 2. This simple model is fully functional, i.e. it is prepared for numerical calculation of fluid flow and heat transfer, can be read into CFD solver, i.e. Fluent, and the boundary conditions and models are set.

The procedure can be illustrated on a simple case of a model with only one plate with zero thickness of the plate material. In that case, the vertical coordinate of nodes on plate were $z_n = 0$, the coordinates of nodes on boundary conditions parallel to the plate were $z_n = \pm 0.001$ (m) and the rest nodes had $z_n \in \pm(0, 0.001)$. In the next step, the computational mesh is mapped and coordinates of all nodes x, y, z are read. Then we calculate relative coordinate ψ_n defined as ratio $\psi_n = z_n / 0.001$ for each node. The own method of creating the realistic model is based on dynamic mesh which is provided by software Fluent. Dynamic mesh means that the computational mesh can be altered between two time steps of the calculation. User defined deforming provided by Fluent was used in our case, because it allows to control the movement of each node of computational domain separately. To do that, a user defined function (UDF) must be written and compiled, which is the most complicated step. The shape of the plate, i.e. undulations, is described by functions or by list of coordinates. The general form of functions are $Z = f(x, y)$, where $Z \in \langle -1, 1 \rangle$ and $f(x, y)$ is an arbitrary function.

The final step is deforming the mesh and thus creating realistic model of a real heat exchanger. Nodes are moved only in direction perpendicular to the plate, i.e. only vertical coordinates z_n of nodes are changed. Again, for the simplest case with only one plate with zero thickness, the new coordinate of each node depends on its coordinates ψ_n, x_n, y_n according relation

$$z = \frac{p}{2}\left[Z + \psi_n - Z \frac{\psi_n^2}{|\psi_n|} \right], \tag{1}$$

where p (m) is spacing of plates. Illustration of initial and final deformed mesh is visible in Figure 3.

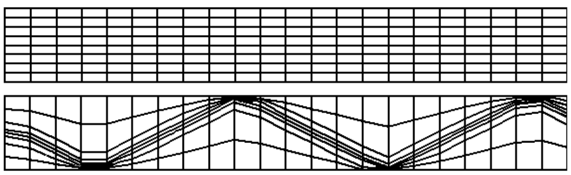

Figure 3. Illustration of the method. Upper – cross section of initial mesh, lower – cross section of deformed mesh.

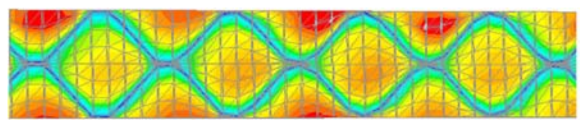

Figure 4. Tight connection of two plates is impossible.

As it follows from description, meshes were created by "pulling" in the direction perpendicular to the plates. Thus, we can create models of most commonly produced plates which are manufactured by pressing or vacuum forming. The main advantages of this method are:

Used mesh is structured and of high quality. It is possible to control height of wall adjacent cells, see Figure 3, which is crucial for correct calculation of heat transfer and flow in boundary layers. All computational variants have similar meshes. Structure of the mesh and count of cells do not depend on the plate geometry. It reduces numerical errors and different variants of plate geometry can be easily compared with lower uncertainty. It is possible to create various topologies, simulate effects of plate spacing, material thickness and others. Generation of geometry is automatic and controlled by in-house software. It allows faster production of optimization variants of the heat transfer surface. The geometry can be change every time step and that creates an ideal tool for optimization. The disadvantages of this method are:

It is impossibility to create zero thickness gap for tight connection of two plates, i.e. there must be a minimal space between two walls. Usually, we use narrow gaps of 0.01 mm without problems with convergence, while the common plate pitch is about 2 mm, see Figure 4 for illustration. Accuracy and reproducibility of details and smoothness of surface strongly depends on mesh density, because local densification of mesh is not possible or is limited. Therefore, more detailed and bigger mesh is required, see Figure 2 to observe roughness of a wavy surface.

Results

The method was used to calculate flow in an air to air plate heat exchanger for HVAC applications, see Figure 6. The plate pitch was 2.2 mm and the thickness of material was 0.08 mm. Reynolds number in the middle part of the plate was about 600 and about 1000 in inlet parts with cross flow. used turbulence model kwSST, which is suitable for calculation such type of flow and heat transfer. Boundary conditions and construction of the model are visible in Figure 5 and Figure 6. Figure 7 presents results of numerical calculation for different meshes. The results are presented in Nusselt number – loss coefficient diagram to show influence of cell size and count of layers of computational cells on heat transfer (Nusselt number) and pressure drop (loss coefficient). Five cell sizes were investigated – 2.0, 1.4, 1.0, 0.7 and 0.5 mm and eight counts of layers of computational cells – 3 to 10, while count m means that there were $2m$ cells across the channel between two plates. The overall count of cells is approximately double for next smaller cell size, i.e. for the cell size 2.0 mm and $m = 3$, the mesh had $N = 743 \cdot 10^3$ cells, while it was $N = 11.8 \cdot 10^6$ cells of size of 0.5 mm and $m = 3$. It is obvious that the computational demands grow with the same pace. The height of wall adjacent cells was 0.1 mm, which correspond to $y^+ \approx 1$ for all cases. This value well satisfies requirements of used turbulence model. As we can see from Figure 7, data obtained from numerical calculations strongly depend on both cell size and count of layers. The highest difference in results are 12.4% for Nusselt number and 13.6% for loss coefficient. Generally we can state that smaller size of computational cells yields higher pressure loss and higher Nusselt number. However, generally these trends depend also on count of layers. We can get results closer to experiments for smaller mesh than for a bigger one.

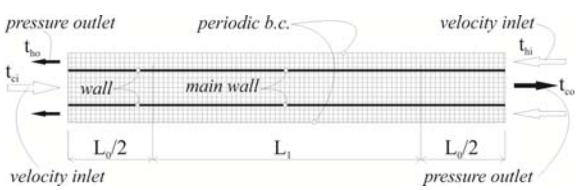

Figure 5. Model of recuperative heat exchanger – boundary conditions.

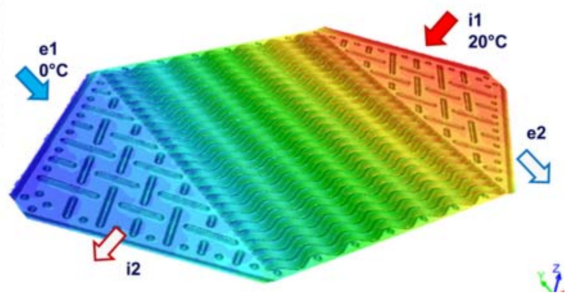

Figure 6. Investigated air to air plate heat exchanger.

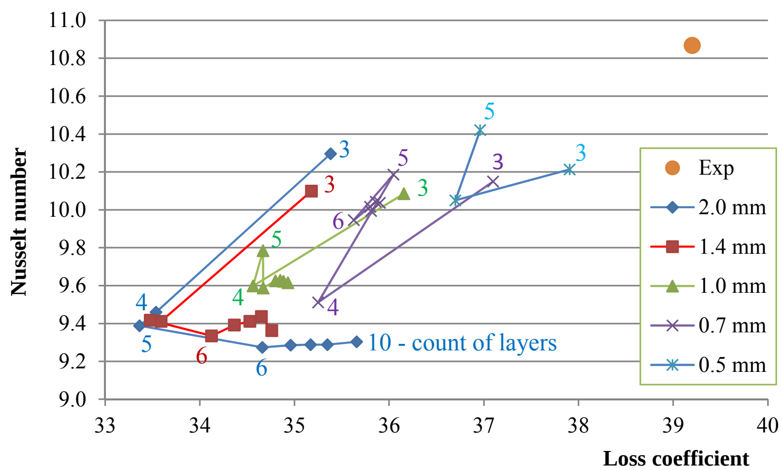

Figure 7. Results of numerical investigation of plate heat exchanger, influence of cell size and count of layers.

We can assume from results for different count of layers of computational cells that the results become more relevant for higher count of layers. Heat transfer and loss coefficient are strongly overestimated for too small count of layers, $m = 3$, and are reduced for higher values of m. It seems that at least $m = 6$, i.e. 12 layers of computational cells across the channel, are required to describe flow in plate heat exchanger correctly. We can see that there is still great difference between results obtained with smallest cells, i.e. 0.5 mm and bigger cells of 0.7 mm. It indicates that the mesh should be even bigger with smaller cells. The problem is that mesh with cell size 0.35 mm and count of layers $m = 6$ would have about 44 million cells. That task would need computer with minimum of 48 GB of RAM memory, which is feasible but no longer easily applicable for optimization. Note that investigated heat exchanger is one of the smallest produced. However, even for smaller meshes we can obtain relevant data for optimization. A difference is also between numerical results from the most detailed model and experimental investigation. The experiments were carried out on a real heat exchanger with over 100 plates. The difference again indicates that used cells were not small enough.

There can be also difference between the geometry of the real plate, which can be deformed due to over pressure or due to manufacturing process, and the geometry of numerical model, which can be altered due to conversion process.

Conclusions

A new method of fast creation of meshes of computational models for computing of flow and heat transfer in plate heat exchangers was presented. The own method of creating the realistic model is based on dynamic mesh which is provided by software Fluent. User defined deforming method was used in our case, because it allows to control the movement of each node of computational domain separately. To do that, a user defined function (UDF) must be written and compiled, which is the most complicated step. The shape of the plate is described by functions or by list of coordinates. Nodes of the mesh are moved in the direction perpendicular to the plates. Thus, we can create models of most commonly produced plates which are manufactured by pressing or vacuum forming. We used this method to investigate flow and heat transfer in a plate heat exchanger of air to air type. We investigated influences of cell size and count of layers of computational cells on heat transfer (Nusselt number) and pressure drop (loss coefficient). We found that data obtained from numerical calculations strongly depended on both cell size and count of layers. The highest difference in results were 12.4% for Nusselt number and 13.6% for loss coefficient. Generally the higher mesh density yields higher pressure loss and higher Nusselt number. Heat transfer and loss coefficient were strongly overestimated for too small count of layers and were reduced for higher counts. It was seems that at least 6 layers, i.e. 12 layers of computational cells across the channel between two plates, are required to describe flow in plate heat exchanger correctly. There were still great differences between experiments and results obtained numerically with model with smallest cells. It was proved that an applicable method was developed to simulate fluid flow and heat transfer in plate heat exchangers. This method will be the base for next investigation and optimization for both air to air and liquid plate heat exchangers.

Acknowledgement

Author gratefully acknowledges financial support by Czech Technological Agency under the project TACR TA01020313.

References

[1] I. Gherasim, N. Galanis, C.T. Nguyen, Heat transfer and fluid flow in a plate heat exchanger. Part II: Assessment of laminar and two-equation turbulent models, International Journal of Thermal Sciences 50, 2011, 1499-1511.

[2] Y. Ch. Tsai, F.B. Liu, P.T. Shen, Investigation of effect of oblique ridges on heat transfer in plate heat exchangers, International Communications in Heat and Mass Transfer 36, 2009, 574–578.

[3] F.B. Liu, Y. Ch. Tsai, An experimental and numerical investigation of fluid flow in a cross-corrugated channel, Heat Mass Transfer 46, 2010, 585–593

[4] J. Novosád, V. Dvořák, Investigation of effect of oblique ridges on heat transfer in plate heat exchangers, Experimental Fluid Mechanics 2013, November 19.-22., 2013, 510 - 514.

[5] J. Novosád, V. Dvořák, Numerical investigation of counter flow heat exchanger with undulation in herringbone patter, International Conference on Sustainability in Energy and Buildings 2015 (SEB 2015), July 1.-3., 2015.