FLOW SIMULATION AND CONJUGATE HEAT TRANSFER IN A PLATE HEAT EXCHANGER

Michael Kröger,Wolfgang Ottow
(ESI Software GmbH, Germany)

1. Introduction

Efficient ways of transferring energy in HVAC systems have been and still are a common need in industrial and private buildings. Extensive research has been done in this area. The development of new technologies opens up more ways to improvements. Among the most prospective of these, is the usage of nano technology, either in form of nano-structured surfaces or in form of nano-particles used in refrigerants. To estimate the efficiency of these approaches, simulations are used to accompany experiments in lab scale as well as in large scale up to industrial size. In the scope of the EnE-HVAC-project funded by the European Commission, the flow and conjugate heat transfer in a real size heat exchanger is investigated. The investigation focuses on identifying parameters that can be used as means to model the influence of nano-structures or nano-particles used in heat exchangers of this type. Studies have shown, that surface modifications can significantly enhance the heat transfer through a thermal boundary, especially under phase change conditions[1]. One possible modification of such a surface is to use nano-structures. This can be done by removing material, e.g. with a laser, or by applying nano-particles which are attached to the surface by physical deposition followed by a curing or sintering process. These modified surfaces can subsequently be tested in a laboratory for their effect on the heat transfer. Testing them in industrial scale requires a lot of effort. In the EnE-HVAC project, modified nano-structures are developed and tested. Simultaneously, a simulation method is developed which allows industrial size simulation without extensive experiments in this scale. This method and a case study is presented in this paper.
The CFD simulations in this paper are carried out with the CFD tool CFD-ACE+.

2. Mesh generation and model setup

For the simulation described here, a plate heat exchanger from the project partner Vahterus is used. It consists of a number of riffled plates, that are welded together.

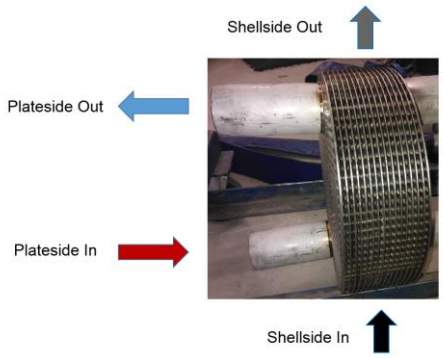

Fig. 1: Plate heat exchanger without shell encasing.

At the front closing plate there are the short pieces of the inlet and outlet connecting pipes for one fluid (plate side), while the short pieces of the inlet and outlet connecting pipes for the second fluid are on top and bottom of the encasing shell (shell side). An image of such a heat exchanger is shown in Fig. 1

The CAD-Modell, cf. Fig. 2 of this heat exchanger is meshed using automatic meshing algorithms with the meshing tool CFD-VisCart.

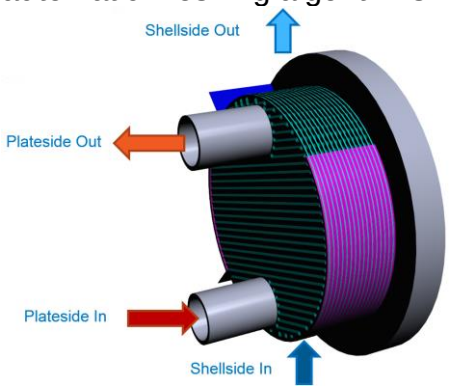

Fig. 2: CAD-Model of plate heat exchanger

The multi-domain grid generation process is based on an interior to boundary approach, which also allows the meshing of non-watertight geometries. The interior volume grid must be generated first and then the interior volume grid is connected with the boundaries. A schematic of this grid generation technique is shown in Fig. 3.

The boundary surface grid is hence a result of the grid generation process and not a prerequisite. A significant advantage is that CAD cleaning is no longer required. This reduces the grid generation time by several orders of magnitude and renders the process much more reliable.

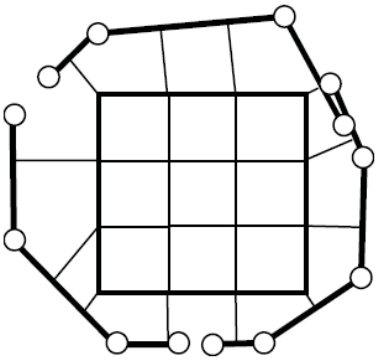

Fig. 3: Schematic of centre to boundary grid generation

With respect to the Vahterus Plate & Shell heat exchanger that was used here it is mandatory to resolve both the two fluid domains of the plate side flow and the shell side flow as well as all solid domains of the heat exchanging individual plates separating the two fluid domains.

Inside CFD-VisCART the individual domains have been defined by positioning so called domain markers at arbitrary positions inside the individual domains. With the 32 plates and the 2 fluid domains this results in the definition of 34 domains. The solid parts of the shell and other enclosing parts are not resolved as it is assumed that the heat transfer to the environment could be neglected.
Furthermore proper mesh size settings and the definition of certain sources for mesh refinement in certain regions have been defined. For the 0.7mm thick plates a cell size of about 0.2mm was used while other regions are resolved with 1 to 5 mm cells.

The actual mesh generation and the grid quality improvement processes have been pursued by running a python script without using a graphical user interface. The following commands are done in background without a graphical user interface:

1. Loading of the model set-up file
2. Automatic mesh generation
3. Grid quality check and automatic improvement
4. Writing of the mesh file
5. Automatic decomposition for parallel computing
6. Writing the CFD-ACE+ model file

The total meshing process and writing of the model files took about 2 hours. The resulting mesh has 48 million cells

Fig. 4: Final mesh of plate heat exchanger

The definition of all model settings could be done in the graphical user interface of CFD-ACE+. To avoid long loading times a simplified model was used to define all needed settings like thermodynamically properties, boundary conditions and solver settings. Based on that model a python script was automatically generated by CFD-ACE+ and slightly adapted to work for the actual large model. This script was then used to set-up the final model in background without running a graphical user interface.

The solver run was conducted on a standard Linux Cluster using 80 cores. Approximate run time was about 6 hours.

The post-processing was again conducted using a python script.

The application of python scripts for all the different stages of the CFD simulation gives the opportunity to easily run several parameter studies depending on either geometrical or process parameter.

3. Solution Method

In this paper the finite volume method is used to solve the incompressible Navier-Stokes equations for mass and momentum. Heat transfer processes are computed by solving the equation for the conservation of energy. CFD-ACE+ numerically solves the energy equation in the form known as the total enthalpy equation. This form is fully conservative and is given in equation.

$$\frac{\partial(\rho h_0)}{\partial t} + \nabla \cdot (\rho \mathbf{u} h_0) = \frac{\partial p}{\partial t} + (\boldsymbol{\tau} \cdot \nabla)\mathbf{u} + S_h + \nabla \cdot (k_{eff} \nabla T)$$

In this equation, the fluid density is denoted by ρ. **u** is the velocity vector, *t* is the time and ***T*** the temperature. Symbol *p* describes the pressure and *u, v and w* are the components of the velocity vector. *t* is viscous stress tensor. In this equation k_{eff} is the effective heat conductivity which is calculated as

$$k_{eff} = k_{lam} + \frac{\mu_{turb}}{\mu_{lam}} k_{lam} + k_{surf}$$

where k_{lam} is the laminar heat conductivity and μ_{turb} and μ_{lam} are the turbulent and laminar viscosities. The last term k_{surf} is a parameter that

models the influence of the structured boundary and is only applied to the cell layer next to the thermal boundary. The value for k_{surf} can be determined either by lab scale experiments of small samples of the modified surface or by detailed direct simulations in the same scale. S_h stands for additional source terms to account for heat sources due to chemical reactions, radiation etc..
The CFD-Solver uses an algebraic multigrid solver to solve the linear equations. More details on the solution method can be found in the CFD-ACE+ user manual[2]

4. Results

The first result is the detailed velocity field for both fluid domains. Figure 4 shows the velocity magnitude in a small portion of the heat

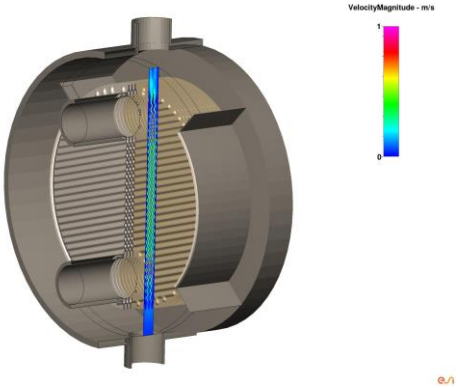

Fig. 5: Flow field in part of a heat exchanger

exchanger. By examination of various parts of the model the manufacturer can identify problematic zones in the assembled heat exchanger which are inaccessible without simulations.

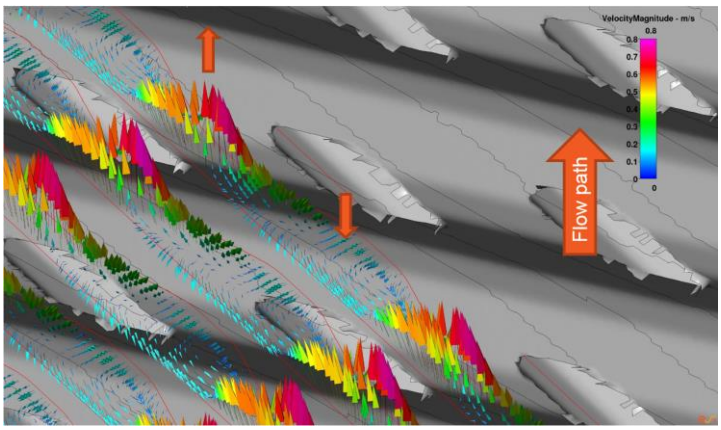

Fig. 6: Detailed flow field in heat exchanger.

In Fig. 7 the pressure distribution is shown. The predicted pressure drop is 10000 Pa, unfortunately no exact measurements are available to verify this result.

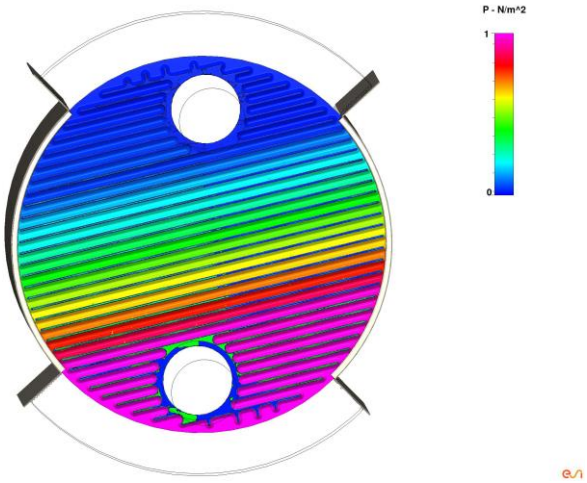

Fig. 7: Pressure field in plate heat exchanger.

After the flow field has been determined, the heat transfer was simulated. The temperature at both inlets and at both outlets has been measured and the results are compared to the simulated values. The plate side values are predicted with an error of 2.5 K while the shell side values are predicted with an error of 0.4 K. The margin of error for the experiments is not known to the authors.

Table 1: Comparison of temperature results from simulation and experiment.

	Experiment		Simulation	Error
	T_{inlet} [K]	T_{outlet} [K]	T_{outlet} [K]	[K]
Plate side	343.15	322.0	324.5	+2.5
Shell side	293.15	314.5	314.1	-0.4

The simulated distribution of the temperature is shown in *Fig. 8*. This detailed distribution gives a lot of insight of how the heat exchanger design could be improved.

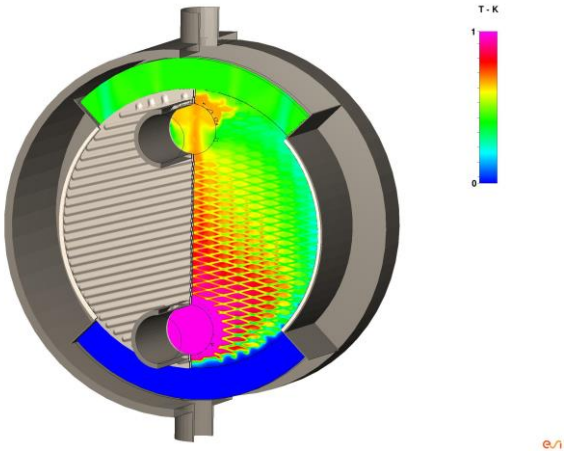

Fig. 8: Temperature distribution in plate heat exchanger

The same simulations where repeated with a modified k_{eff} value to mimic the effect of a nano-structured wall. In preliminary studies it could be shown that, by changing the heat conductivity in just on layer of cells at the boundary, the overall heat transfer from the wall to the fluid can be influenced. This is qualitatively demonstrated in Fig. 9. Here, two simple straight channels are used. The dividing wall is a thermal boundary with fixed heat flux. In the left channel, the standard heat conductivity is applied. In the right channel, an increased k_{eff} leads to a higher heat transfer.

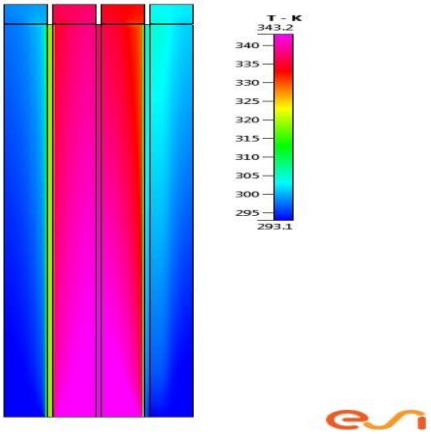

Fig. 9: Temperature distribution with standard k (left) and modified k (right)

This works also well in case of the large scale simulation where an increase of k_{eff} leads to a significant decrease of the outlet temperature at the shell side.

5. Conclusion

The possibility of k_{eff} as a model for small scale boundary effects like nano structuring in large scale CFD simulations has been tested and verified for the case of a plate heat exchanger. This is only a qualitative result right now, since experimental data is not complete for the modified structure in the real heat exchanger, yet. Future simulations will determine the exact value of the modified heat conductivity. It is

assume that under certain conditions, e.g. phase change, a fixed value for k_{eff} has to be replaced by a function depending to the local velocity and or temperature.

6. Acknowledgement:

This project is funded by the European Commission under the title „EnE-HVAC“.
The authors would like to thank project partners, Vahterus, Finnland and the Danish Technological Institute (DTI), Denmark for providing experimental data for the investigated heat exchanger. Vahterus is a leading manufacturer of plate & shell heat exchangers used in oil and gas, chemical and process, energy and refrigeration industries. The Danish Technological Institute is a self-owned and non-profit research institution.

7. References

[1] Kim,H.(2011): *Enhancement of critical heat flux in nucleate boiling of nanofluids: a state-of-art review*, Nanoscale Research Letters, 6,pp. 415-433

[2]] CFD-Ace+ Manual, ESI-Group, Version 2014.0, **2014**