Czech Technical University in Prague, Faculty of Mechanical Engineering
Department of Process Engineering

Review of Master Thesis

Author: **Mehmet AYAS**

Title: **Model of the Plate Heat Exchanger**

Specialization: (N2301) Mechanical Engineering

Filed of study: (3909T012) Process Engineering

The thesis has focused on designing a model of plate heat exchanger in software ANSYS, performing a simulation for given operational parameters, and comparing the results of simulation with experimental data.

Formal comments and mistakes

There are many formal mistakes in the thesis, some of the are mentioned below.

Page 9 – the first line reads "Heat exchanger is a device that is used for transfer thermal energy from one liquid to another". It is probably more correct to say "fluid" instead of "liquid".

Page 11 – the literature source should be added to the captions of Figures 1 a 2. It is also not clear why the caption texts of all figures are enclosed in parentheses.

Page 12 – bar is used in one part of the text as pressure units, I would recommend to use same units as in the following text (Pa, MPa).

Page 29, section 3.1 – it seems that the whole text of the first paragraph (at least) was copied from the www.ansys.com, so some reference should be mentioned here.

Page 34 – it is not possible to read the axis description of Figures 19 and 20.

Page 37 – port diameter 0.021 mm should be probably 0.021 m.

Page 54 – it is more important to know the maximum value of skewness because we usually want to avoid mesh elements with large skewness. Therefore the mentioned value 8.14×10^{-6} is useless because it represents the minimum.

Page 55 – In the part describing the second mesh, the number of mesh elements mentions is 816 764, but it is 574 719 in the Table 17.

Page 63 – reference [13] is not complete, year is missing there, at least.

More serious comments and mistakes

According to the thesis tasks and statement "main aim of this thesis is numerical simulation" at page 36, I would expect more information about the settings of the simulation software (ANSYS Fluent), for example which variant of $k - \varepsilon$ model was used in the simulations, what boundary conditions were used on the wall, what solution procedures were set in the Fluent, etc.

When inspecting the ANSYS project files on the supplementary DVD, it seems that first order discretization schemes were used for turbulent quantities like the turbulent kinetic energy and dissipation rate. Also, only 250 iterations seems to be very small number to be sure that you get a converged solution. Some of the residuals are not below values 10^{-3} which is recommended. Have you monitored some other quantity to verify that the solution is really converged?

It is mentioned on page 56 that standard wall function was used – it is quite incorrect approach with more or less complex geometry. How do you know that you are within a valid range where the standard wall functions can be used?

The $k - \varepsilon$ model with standard wall functions is probably not suitable for this kind of problem where some separation zones and maybe secondary flow patterns could occur. At least, a comparison with some other turbulence model (SST $k - \omega$, for example) should be made, and for heat transfer problems, it is strongly recommended to create inflation layers near walls so that the wall functions are not needed to approximate the boundary layer.

Only two mesh sizes are not sufficient to correctly estimate the accuracy of numerical solution. In some regions of the geometry, there is only one mesh element accross the channel gap – this is really not sufficient and could be one of the reasons why the results differ from experimental data so much and that the "less dense mesh", probably by chance, gives better results as mentioned in conclusions on page 61.

Evaluation: satisfactory (D)

Questions

- Did you verify that you can use the standard wall functions in your situation? What is the quantity which can be used to do this and what values have you observed in your model.
- Can you explain why more than two mesh sizes are needed for an accuracy estimation of some numerical solution?

Prague, 6.8.2015 Ing. Karel PETERA, Ph.D.