Evaluation of the diploma thesis written by Kaan Ege Temiz

Submitted diploma thesis with name „Optimization of heat transfer between fluid stream and heat transfer surface“ was written by Kaan Ege Temiz. The thesis is mainly focused on CFD simulation of some selected geometry using ANSYS CFD program. The work is logically divided into several parts. First chapters of the thesis (Introduction, and surface modification techniques) deal with introduction into techniques which can be used for improving of a heat transfer coefficient through modification of surface shape. More detail focus is taken on dimple technology. Only this femto part is focused on classical literature survey, but without detailed information. Next part deals with description of used procedures and software during simulation of wall jet and impinging jet heat transfer. This part consist of geometry description of the subject, description of CFD solver, results, and their discussion. The results are presented using graphs (local values of heat transfer coefficients) and tables (average heat transfer coefficient values) without additional parameters such as size, averaging area, ... The whole work is then completed with an evaluation of the performance evaluation criteria for selected geometrical arrangements.

The thesis is interesting, not too long, but with many formal and factual errors and inaccuracies.

- The list of symbols is missing. There are unnumbered objects (figures) in this work.
- Wrong use of citations. A lot of the text is without proper citation.
- Bad literature survey, or proper literature survey is missing. You have to find, rewrite, ... specific information like correlations, equations, graphs, tables from literature, i.e. information which demonstrates results of object of your study. Only a few paragraphs on page 23/24 look like nucleus of literature survey.
- Description, and especially size, of numerical domain, is missing (it is based only on scale inside bitmap pictures taken from Ansys Fluent), but information about size of used corrugation of heat transfer surface (width, height, depth) is missing whatsoever.
- There is not heat transfer coefficient definition in your work. How can we evaluate heat transfer coefficient using Fluent solver? Which parameter we have to set during this evaluation?
- It would be great to include all parameters, sizes, ... used during calculation, along with intermediate results (temperatures, dimensionless numbers, ...).

The work is typical work focused on CFD. Author shows his experience with creation of numerical model of real system using ANSYS CFD system, with numerical solving and interpretation of results. The thesis is not so long. The thesis is written by simple and rather clear form, and contains some formal and factual errors and inaccuracies. Some of them I marked directly in the text. I am not able to evaluate language purity of this work. Author accomplished the given tasks. With respect to the reasons above, I recommend the thesis to defense with evaluation

C (good)

and I am pleasured, that I can ask author about several questions.

↪ You wrote that „Researches show that baffles/ribs/fins/dimples have been . . . “. What is the difference between rib and fin?

↪ Why did you solve problems in 2D? If you want to solve heat transfer with spherical dimples, therefore, I think that you cannot use 2D dimples. Two dimensional dimples are like cylindrical dimples. Are the results of 3D spherical dimples and 2D cylindrical dimples comparable? Why didn't you use 3D geometry for wall jet simulation? You used only 2D cylindrical dimples in the case of wall jet.

↪ You used constant heat flux as a boundary condition during your simulations. Is it real boundary condition? How can we implement this boundary condition in real apparatus? How will the results of heat transfer simulation be influenced by changing boundary condition to constant temperature condition?

↪ What characteristic length did you use during calculation of the Nusselt number using Equation 6?

↪ Why didn't you try to compare your results, e.g. local and integral values, with results from literature? There are a lot of articles dealing with heat transfer on dimpled surface, especially for impinging jet heat transfer.

Martin Dostál
m. p.
Department of Process Engineering
Faculty of Mechanical Engineering
Czech Technical University in Prague

Prague, 22nd January 2022