

Review of the master thesis with title *CFD simulation of heat transfer in an agitated vessel*

The thesis, written by Arunanshu Chakravarty, is primarily focused on the application of the commercial numerical solver ANSYS CFD for solution of the flow field and transient heat transfer in a cylindrical vessel agitated by a pitched blade turbine impeller. The thesis is divided into several logical parts. Introductory parts deal with description of a standard configuration of an agitated vessel and with the dimensionless criteria usually used in heat transfer theory. Introductory parts are closed by a brief theory used in turbulence modelling, i.e. the balance equations and by description of the RANS models. In the next parts author describes typical sequential process of CFD modelling, i.e. description of the model geometry, description of the computational grid and its refinement and processing of the calculated results. Author uses in this work the Sliding Mesh Technique. The author puts great emphasis on data postprocessing, i.e. the power number evaluation and the heat transfer coefficient evaluation and he states several methods how to do it.

The thesis is written by simple and rather clear form. The thesis fulfils the assignment and contains not so much formal and factual errors and inaccuracies (some of them have been highlighted directly in the work). With respect to the reasons above, **I recommend the thesis to defense with evaluation B (very good).**

I have only the following questions or recommendations to the author.

- „Make a literature research . . . “ usually means working with literature. In the list, there are 23 sources but in thesis I have found hardly one half of references. In your thesis there are large parts, e.g. theoretical part, without any reference.
- Please, explain formulation „The transition (to turbulent flow) also depends on the power number“, see page 8.
- Did you try to compare the dimensionless axial velocity profile on figure 6.3 with some results following from literature survey?
- Did you find some results justify your power model of the power number in turbulent region?
- What do you think? Is it possible to apply the logarithmic mean temperature difference along the time axis? Do you have some proof of this usage?
- You have described three methods of the heat transfer coefficient evaluation from the CFD results. Which approach is the best?

Martin Dostál

m. p.

Department of process engineering
Faculty of mechanical engineering
Czech technical university in Prague

Prague, August 31, 2017