Study program: Mechanical Engineering
Study field: Process Engineering
Author: Gokul Sai Namburi
Title: CFD simulation of heat transfer in an agitated vessel with a pitched six-blade turbine impeller

Author performed numerical simulations of fluid flow and heat transfer in an agitated vessel with a pitched six-blade turbine impeller in ANSYS Fluent. He used “Sliding Mesh” approach and transient simulations in ANSYS Fluent. To eliminate the influence of temperature increase on the evaluation of heat transfer coefficient, the internal heat source (sink) balancing the heat supply was used. He also investigated the impact of different impeller distances from the bottom. Even though the author spent on the thesis a lot of time, I would expect more independency in his work. Graphical postprocessing and presentation (plots, figures) of the results could be done better. The methods and procedures used in the thesis are correct but the resulting correlations for the Nusselt numbers are not in very good agreement with available experimental data – it is a question for further work what might be the main reason.

Evaluation: satisfactory (D)