Review of master thesis

Alend Sharafani:

CFD simulation of the agitated batch

The thesis is focused on numerical modeling of the flow field in agitated batch using CFD (Computational Fluid Dynamics). The main aim should be (according assignment) to create benchmark task for modeling of the agitated batch, namely in turbulent regime, with LES (Large Eddy Simulation) approach.

The given major points of the thesis were the following:

i) literature search of LES method applied for agitated batch,

ii) preparation of computational model (ANSYS),

iii) mesh optimization,

iv) simulation using LES method,

v) comparison of the calculated and experimental results.

In this general view the submitted thesis satisfies the basic given points. The first part is focused on description of the modeling as well as simply discribes commonly used experimental methods for velocity measurements in stirred tanks. The second part (called “Solution procedure”) deals with model preparation. This part is composed very well, let say in “user friendly” quality in some sections. The third part (called “Fluent results”) represents the calculation results, calculation optimization and comparison with experimental data.

On the other hand the literature search is not disposed quite well (contains twelve items and only three are focused on LES in agitated batch). The better literature search may bring more information about some crucial settings, used turbulent models (their advantages and disadvantages) and more experimental data for comparison. In the first part, there is not clearly discussed a number of other topics e. g. averaging process in calculations (minimal calculation times for averaging), reason for choosing of the turbulent model (k-omega SST, LES Smagorinsky-Lilly), reason for choosing of the used calculation schemes and their effect on the calculations accuracy.

Some very important parameters (not only) for LES simulations were not highlighted and analyzed e. g. wall treatment (y^+ conditions, wall functions), CFL condition (Courant number), analysis of mesh size independence etc.

The comparison of velocity profiles is uncleared. The cited article did not contain the profiles of 4-PBT, only results for 6-PBT. The table 6 compares experimental vs. calculated data of impeller power number, but literature source is not cited.

Some minor remarks:

On page 24 in subsection “Pre processing”: All values are mismatched. What means expresion: “agitate vessel with speed 300 $m.s^{-1}$ “ supersonic regime in the vessel? In my opinion impeller Reynolds number was Re = 50 000 not 500 000 and comparison with value 3200 was probably exchanged with turbulent regime in pipe flow instead of approximately 10^4 (for constant impeller power number) used for high speed impellers in baffled vessel e. g. sttired by 4-PBT.

Figure 20 on page 41 is something curious without any explanation... “lost figure”?

From the general point of view, the work meet the rquirements of the master thesis assignment and with respect to above mentioned comments I suggest this work to defense with grade

good (C)

Questions for the defense:

1) How the wall functions in RANS model k-omega SST were setup. Explain the right settings of mesh size near the wall for both RANS or LES in relation with $U^+=fce(y^+)$. What recomens the FLUENT user's guide?

2) Why the "pressure based" solver was used.

In Prague, August 10th, 2015 Ing. Bohuš Kysela, Ph.D.