CZECH TECHNICAL UNIVERSITY IN PRAGUE

Faculty of Mechanical Engineering, Department of Process Engineering

Review of Diploma Thesis

Study program: Mechanical Engineering

Study field: Process Engineering

Author: **José Renato Ronzon Tirado**

Title: **Investigation of the particle movement in rotary kiln**

The thesis focused on CFD simulation of particles in a rotary kiln using software ANSYS, setting up and performing simulations for given operational parameters, and comparing the results of simulations with literature data.

Formal comments and mistakes

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

Page 4 – N should be called as rotational speed, not angular velocity.

Page 18 – Caption of the table should not be "Figure 5" but "Table 5". These Figure names instead of Tables are used in the whole thesis.

Page 22 – Numbers of figures in their captions do not match the referenced numbers in the text.

Page 28 – It is better (for the reader) to use [Boateng, 2008] as references to literature sources instead of just [3], see for example caption of Figure 17 (?) on page 28 and many other places in the thesis.

Page 31 – It is not obvious what symbol Φ_0 means in the equations here.

Page 47, item 7 – I think that "Because of the limitations of the ANSYS software the gravitational force has to be divided into its components" is not correct. It is not a limitation of ANSYS software, I think, it is due to the created geometry which is aligned with x-axis, not inclined.

Page 48 – The first sentence is difficult to understand what it means. And I suppose the residence times in the table are for a kiln with no inserts, baffles, etc. It is not clear here and it could be mentioned there.

In the Appendix 4, section and figure titles are somehow messed up. For example, on page 127, and on many other places as well.

I also miss there information what approach of the moving mesh was used in the ANSYS Fluent, MRF (Moving Reference Frame) or Sliding Mesh.

Factual comments and mistakes

On page 48, Reynolds number is mentioned to be 650 thousand corresponding to the turbulent regime, but I see that in the ANSYS simulations laminar viscous model was used.

There is no grid independence study, that is some estimation of numerical error due to spatial discretization. Because it is a first attempt of simulating such complex phenomena and the focus was on selecting proper multiphase models and setting up the solver, it can be accepted here.

But I think that more important would be an estimation of errors coming from the choice of the time step. This may be the crucial parameter affecting the resulting residence time.

In the simulations, velocity-inlet boundary condition was used at the zone representing physical outlet of the gas phase. Negative (-1) x-component flow direction were used there (see the top figure at page 134 or 144, for example) to follow the counter-flow arrangement. But this velocity condition represents a constant velocity through the whole zone which is probably not physically correct.

I appreciate that there is a lot of information about settings of FLUENT solver in the thesis, but I miss there information how the residence times were evaluated – did you really used residence times of particles which left the domain through the outlet?

In Simulation 3, I think that the moving mesh was applied to the whole geometry, that is including the inlet of the clay. If you wanted to avoid this, you could slice off some part if the inlet cylindrical section and apply the moving mesh only to the rest of the geometry. And at the inlet section you could just set up a boundary condition of moving (cylindrical) wall.

Even though there are many mistakes in the thesis, the student made a lot of work to be able to understand such complex phenomena, understand and select appropriate multiphase models in ANSYS FLUENT solver, and finally perform the presented simulations. He showed that his abilities and knowledge are on a very good level.

Evaluation: very good (B)

Questions

- According to the given flue gas velocity and evaluated Reynolds number (page 48), turbulence flow regime should exist there. Why the laminar viscous model was used?
- Can you explain in more details how the residence times were evaluated?
- On page 74, you state “The simulation 2 method seems to be more accurate”. What could make the main difference with respect to Simulation 1?

Prague, 24.8.2018 Ing. Karel PETERA, Ph.D.