COMPARISON OF ANSYS FLUENT AND OPENFOAM IN SIMULATION OF CIRCULATING FLUIDIZED BED RISER

Markku Nikku[1*], Payman Jalali[1,2], Timo Hyppänen[1]

[1]*Lappeenranta University of Technology, LUT Energy, PL 20, 53851 Lappeenranta, Finland*

[2] *Department of Physics & Center for Nonlinear and Complex Systems, Duke University, Durham, North Carolina 27708, USA*

*Email: *mnikku@lut.fi*

Abstract – In this work, two CFD codes are utilized in Eulerian-Eulerian modeling of laboratory-scale cold model of CFB. The objective of the work is to compare the accuracy and performance between the utilized codes, which are the commercial code of Ansys Fluent 17.0 and open-source code of OpenFOAM 3.0.1 with the solver twoPhaseEulerFoam. The same mesh and as similar settings as possible are utilized in transient simulations. The average profiles of pressure, velocity and solids volume fraction are similar between the CFD codes and match well with the measurements from the cold model. The computational times and performance are compared and OpenFOAM was found to perform better, but having more limited time step size.

INTRODUCTION

An increased computational capacity in modern computers has led to a boost in utilization of different modeling tools in simulations of various apparatus' and systems. The modeling is an inexpensive option to test new physical concepts as well as designs, rather than building prototypes of large and high-cost devices. Considering circulating fluidized bed reactors and furnaces, where multiphase flow, heat transfer and thermochemical reactions can occur simultaneously with significant intercoupling, it is not either easy to perform extensive measurements of such system, or even feasible in large scale. Thus, modeling can provide more insight on the behavior of a larger system with limited measurement possibilities. Reliable modeling of such a complex system requires the utilization of validated and tested methods and models. For the model, the validation measurements are crucial.

Panday et al 2014 presented results of blind modeling of NETL laboratory CFD unit with the height of 15 m and diameter of 0.3 m. The simulations were performed by the Eulerian-Eulerian approach in 2D and 3D with coarse and fine meshes, as well as 3D Eulerian-Lagrangian with two different particle counts. By the comparison of simulation results with measurements, it was uncertain to conclude what method or settings would result the most accurate prediction, as all of them showed large variations from the measurements. Zhong et al 2015 simulated a laboratory CFB unit (height 1.8 m, diameter 0.125 m) with 3D Eulerian-Lagrangian method and included the reactions representing combustion. Wang et al 2015 utilized Barracuda VR in 3D simulation of a CFB unit (height 3.0 m, diameter 0.40 m). They compared their results with 2D Eulerian-Eulerian simulations by Niemi 2012, which utilized a particle size distribution, and reported better accuracy and lower demand for computational resources. The computational times in MP-PIC method increased from approximately 190 to 400 hours by increasing the number of parcels from 400.000 to 890.000 for 30 seconds of real time. Zhang et al 2016 utilized MP-PIC method in pseudo-3D simulation of two CFB risers with MFIX with particle size distributions. Zeneli et al 2015 utilized Ansys Fluent with EMMS approach in 3D modeling of CFB carbonator pilot (height 8.661 m, diameter 0.59 m) with limestone reactions. Their computational times varied from 15.4 to 333.35 h for 20 seconds of real time.

There are a number of studies presented in the literature comparing different codes; e.g. López et al 2015 compared Ansys Fluent 15 and OpenFOAM 2.2.x with Lagrangian-Eulerian jet impingement erosion, Balogh et al 2012 compared Ansys Fluent 13 with OpenFOAM in study of atmospheric boundary layer, and Herzog et al 2012 compared Ansys Fluent 6.3, MFIX and OpenFOAM 2.0 in modeling of bubbling fluidized bed.

Ansys Fluent has become almost synonymous with CFD over the years. The software package contains several features and the basic use is quite easy to learn. The documentation has been utilized in several scientific journal articles as a reference, as some of the models are easier to find in it than in the original articles. The software has two licensing options, commercial and academic, which comes at a lower cost for academic users. On the other hand, the software licensing comes with a price, for example parallel computing needs one additional purchased parallel license after a certain number of parallel cores provided by the basic package. This can lead to cost factors limiting the number of utilizable licenses and therefore cores in massive parallel computations.

The main attractiveness of OpenFOAM is that it is open source software. It is cost-free, the source code is available for development, and there are no licensing limitations for example on parallel computations. The main restrictions include steep learning curve, limited available documentation, and a requirement of understanding the physics, numerical methods and programming in C++ to be able to implement own contributions.

In this work, two CFD codes are utilized for Eulerian-Eulerian modeling of laboratory-scale cold model of CFB. The objective of the work is to compare the accuracy and performance between the utilized codes from the user's point of view. This refers to not extensively studying the inner workings of the simulation software, but focusing more on performing the simulations. The utilized codes are the commercial Ansys Fluent 17.0 (Ansys 2016) and the open-source code of OpenFOAM 3.0.1 (OpenFOAM foundation 2015) with the solver twoPhaseEulerFoam (Rusche 2002, Peltola 2009). Identical computational mesh was created for the CFB cold model riser using Ansys ICEM CFD (Ansys 2016). The models and boundary conditions were set similarly between the CFD codes. Transient two-phase flow simulations were performed in 3D using parallel-processing computation on identical processors. Averaged values are calculated over time and space. The average profiles of pressure, velocity and solids volume fraction are compared between the CFD codes as well as the available measurements from the laboratory device of cold model CFB. Additionally, the computational times and performance are compared.

EXPERIMENTAL METHODS

Measurements were conducted in a laboratory scale CFB device in ambient temperature and pressure. A schematic of the device is presented in Fig. 1. The riser has a diameter of 0.11 m with the height of 1.8 m along which several pressure measurement connections are placed. The device is operated with compressed air and mass flow controllers are used to regulate the airflow to the windbox and to the loopseal, and the device is capable of maximum superficial air velocity of 10.5 m/s.

For the validation test, the device was loaded with 5 kg of glass beads, which have the material density of 2465 kg/m^3. The distribution of material remains relatively constant during the experiments with approximately 1.75 kg in the riser and 3.25 kg in the loopseal. Small losses of material, less than 1 % of input mass, were measured after the experiment. The glass beads particle size and shape distributions were determined with a 2D image analysis. The volume average particle size is 490 µm. The glass beads can be considered spherical as their average measured circularity was 0.92, where a circularity of a perfect circle is 1. The sample was fluidized with a constant air flow of 3.25 m/s in the windbox and 0.2 m/s in the loopseal for several minutes, while differential pressure transducers were used to measure the vertical pressure distribution. The errors of the used pressure transducers varies from ±1 Pa in the upper part of the reactor to ±8 Pa in the lower part of the reactor.

After the measurement of the pressure profile, the external circulation mass flow rate was measured. This process involved a pinch valve in the return leg, which is supported by a strain gauge and operated with the compressed air. The pinch valve was closed and mass accumulation of the particles falling from the cyclone was measured in real time with the strain gauges. After collecting approximately 200 g of mass, the valve was opened and the several minutes were allowed to pass before repeating the measurement in total 6 times. From the strain gauge data, the external circulation mass flux was obtained as the time derivative of the mass accumulation.

NUMERICAL METHODS

Eulerian-Eulerian approach was adopted in the simulation of the test device. In this method, the solid phase is considered as continuum and several closures are required to close the mass, momentum and energy conservation equations, namely kinetic theory for granular flow, describing solids pressure, viscosity and granular temperature. Gas phase turbulence was not considered and simulations were performed as laminar. The utilized submodels for both codes are listed in Table 1.

Fig. 1. The laboratory-scale CFB device (left) and the computational mesh (right).

Table 1. List of the utilized submodels. For details, please refer to the given software references.

	OpenFOAM (2015)	Ansys Fluent (2016)
Gas-solid drag	GidaspowErgunWenYu	Gidaspow
Granular viscosity	Gidaspow	Gidaspow
Granular pressure	Lun	Lun et al
Frictional stress	Schaeffer	Schaeffer
Radial distribution	Lun-Savage	Lun et al
Bulk viscosity	Lun et al	Lun et al
Granular temperature	Transport equation	Transport equation
Granular conductivity	Gidaspow	Gidaspow

The riser of the laboratory device was discretized to a structured computational mesh using Ansys ICEM CFD (Ansys 2016). The average length of the side-face of grid cells is 5 mm and the mesh consists of 290.722 hexahedral elements. The mesh size is roughly 10 times the particle size, which is the rule-of-thumb in fluidized bed modeling (Panday et al 2014), therefore no mesh sensitivity study was performed. The simulated mesh is illustrated in Fig. 1.

The simulation parameters and boundary conditions (Table 2) were set to match the experimental values with both CFD codes. Several transient simulations were performed to compare the effect of time step on the results and simulation performance. The residual control was utilized and maximum iterations per time step was 20. For OpenFOAM, Courant number controlled adaptive time stepping (allowed maximum Co is 0.4 and maximum time step is $1e^{-4}$ s) was used for one case, and the fixed time step of $6e^{-5}$ s for another case as larger time steps led to problems in convergence and conservation of quantities. According to the manual of Ansys Fluent, there is no possibility to use adaptive time stepping with Eulerian-Eulerian multiphase simulations (Ansys 2017), therefore cases were run with fixed time steps of $1e^{-4}$ and $6e^{-5}$ s. Similarly due to convergence of OpenFOAM, the residual limit of $1e^{-5}$ was selected.

For recirculation of the particles exiting to the cyclone through the riser exit, a closure was implemented to keep the mass of the system constant. In Fluent a user defined function and in OpenFOAM the Swiss Army Knife for Foam AKA swak4Foam (swak4Foam 2016) were utilized to compute the exiting particle mass flux and implement equal mass flux to the solids return channel boundary.

Each simulation was carried out in a single node of Linux computer cluster. The computational node consists of 2 Intel Xeon E5-2698 with total of 32 cores and 256 GB of memory. Each simulation utilized the full 32 cores and transient operation was simulated for 10 seconds for averaging after the initialization and development of quasi-steady two phase flow. The computational time required for completing the simulations is presented in Table 3.

Table 2. The primary boundary conditions. OpenFOAM information in italics.

	Grid	**Solids return**	**Riser exit**	**Walls**
Gas and solids velocity [m/s]	Velocity inlet / *surfaceNormalFixedValue* 3.25 m/s, 0 m/s	Velocity inlet / *surfaceNormalFixedValue* 0.5 m/s, 0.1 m/s	Pressure outlet / *pressureInletOutletVelocity*	No slip
Solids volume fraction [-]	0	UDF / *swak4FOAM*	*zeroGradient*	*zeroGradient*

RESULTS

Table 3 presents the comparison of computational times for the parallel simulations. Fluent could not converge with the set convergence criteria of $1e^{-5}$ for residuals, leading to maximum number of iterations and long computational times. The same is true for OpenFOAM where only few time steps with adaptive time step converged before the maximum number of iterations was reached. This makes the simulation times comparable. OpenFOAM appears to be significantly faster than Fluent with same time step size in case B.

Table 3. Comparison of computational times.

Case	Time step [s]	Total wall time [h]	Share of converged time steps [%]
Fluent C	$1e^{-4}$	175.2	0
Fluent B	$6e^{-5}$	290.6	0
OpenFOAM B	$6e^{-5}$	176.5	0
OpenFOAM A	adaptive ($\Delta t_{ave} = 6e^{-5}$)	128.8	2.4

Fig. 2. presents both the simulated and measured vertical pressure profiles. Both Fluent and OpenFOAM reach very good agreement with the measured vertical pressure profile. Generally, the profiles are almost identical, with small little differences visible in the bottom and middle of the riser. Both the Fluent and OpenFOAM are within the measurement accuracy of the pressure measurements in the middle and top of the riser, while both overestimating the pressure at the bottom of the riser compared to the lowest two measured value at the heights of 0.05 and 0.35 m.

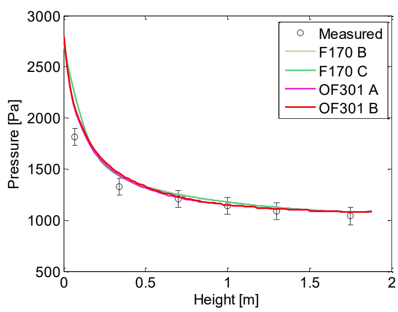

Fig. 2. Comparison of vertical pressure profile with measurements.

Fig. 3 presents the radially and time averaged volume fraction of the solids at different heights in the riser. The radius of 0 m corresponds to the centerline of the riser and the radius of 0.055 m to the wall. Different cases are similar within one code, indicating that the utilized time step does not affect significantly to the results. Comparing Fluent and OpenFOAM, the profiles are quite different. For Fluent, the solid concentrations near the wall are not as large as those in OpenFOAM, but the wall layer is thicker and gradients are smaller. Unfortunately, the wall layer thickness was not measured experimentally which could be used to determine which code prediction is closer to the reality.

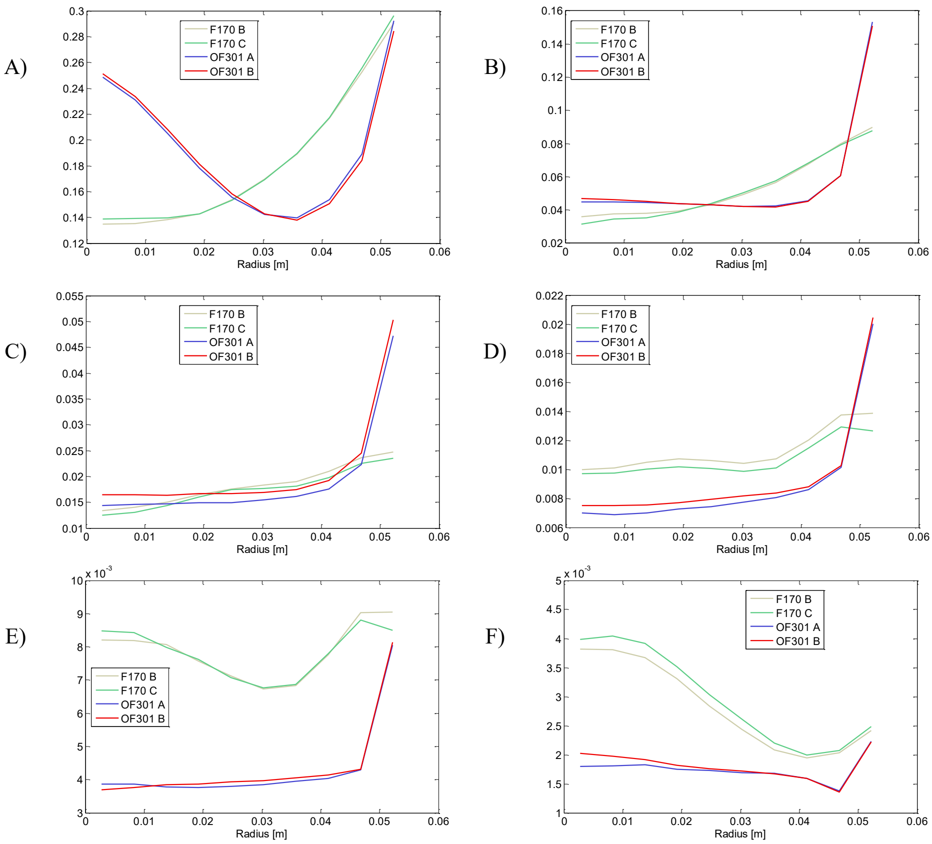

Fig. 3. Comparison of time and radial averaged solids volume fraction.
The heights are A) 0.05, B) 0.35, C) 0.70, D) 1.00, E) 1.35, F) 1.75 m.

Fig. 4 illustrates the radially and time averaged components of air velocity. The lateral components (x and z) are close to zero except at the bottom of the riser due to the effect of solids return, which is located in the vicinity. Fluent also predicts some lateral movement at the height of 0.35 m. The main comparison should be focused to vertical (y) component of velocity. The inlet velocity is 3.25 m/s and due to the effect of wall layer, the gas A) mainly flows in the core region of the riser, and B) has much higher velocity than in the inlet. There is a significant difference in the gas velocities between the codes, which could be explained by differences in solids distributions. The gas flow velocities are higher in the core where the solids volume fraction is the lowest, and they are the lowest on the wall where the solids concentrations are the highest. The gas velocities of OpenFOAM are lower and more uniform in the core due to the lower solids concentrations and sharper gradient near the wall. Similarly, the smaller gradients and thicker wall layer in Fluent results reduce the flow area of the gas and force it to flow faster.

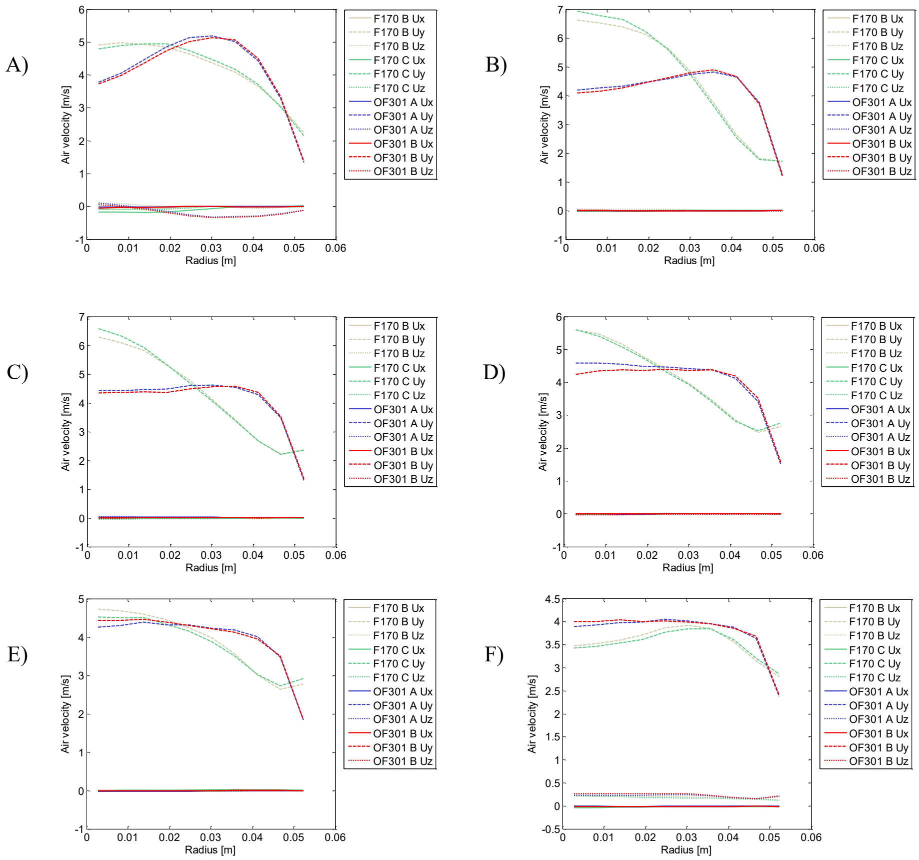

Fig. 4. Comparison of time and radial averaged air velocity. F170 refers to Fluent and OF301 to OpenFOAM, A, B or C states the case, and x, y, or z state the direction of the velocity. The heights are A) 0.05, B) 0.35, C) 0.70, D) 1.00, E) 1.35, F) 1.75 m.

Similar results as presented in Fig. 4 can be seen in Fig. 5 for the radially and time averaged solids velocities. The wall layer and vertical gas velocities explain the higher solids velocities in the core and negative velocities in the wall layer. The differences between predicted distributions are similar to Fig. 4 with higher gas velocity with Fluent causing also higher particle velocities, even double the vertical core velocities of OpenFOAM.

Table 4 presents the measured and modelled particle mass flow rates. The simulated values are average particle mass flow rates at the outlet while the measured values as obtained after the cyclone. The number of particles escaping from the cyclone with the gas flow was very small, thus the correspondence of presented data should be good. It can be seen that the correspondence between the Fluent results and measurements is good. For OpenFOAM, prediction of mass flow rates to the separator is significantly higher compared to the measurements. This due to higher solids mass flux velocity near the riser exit (1.75 m) resulting from higher solids velocity, though the volume fraction is lower than for Fluent.

Table 4. Comparison solids circulation rate.

Particle mass flux to separator	Average measured	Fluent 17.0		OpenFOAM 3.0.1	
		Case C	Case B	Case A	Case B
g/s	13.0 ± 3	14.61	14.62	19.84	20.38

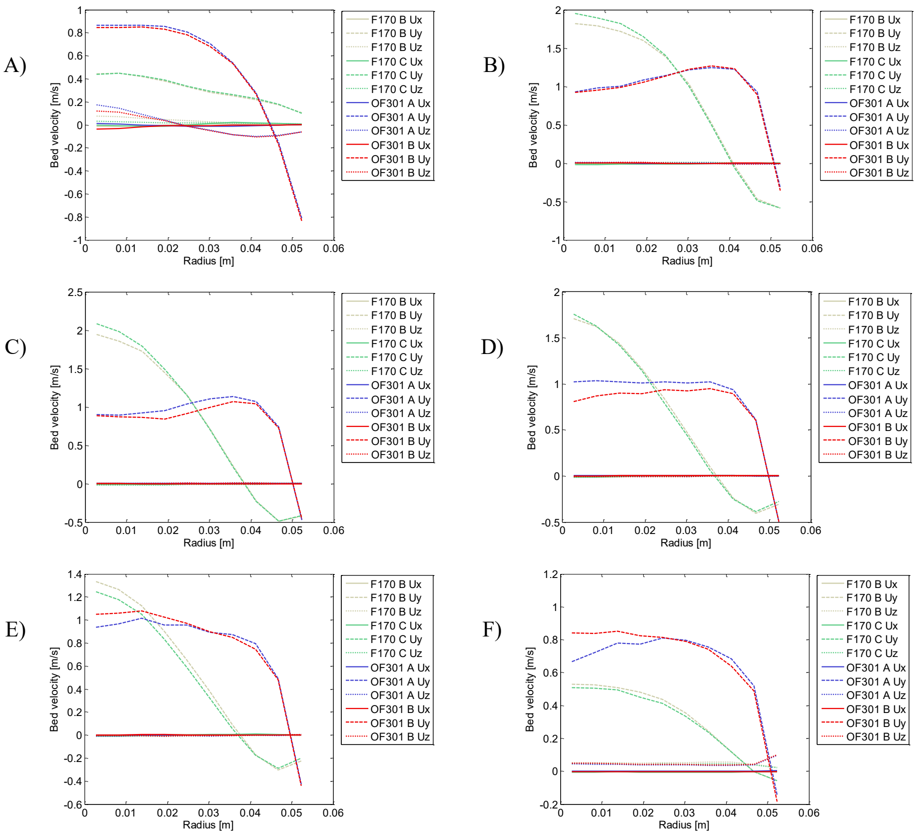

Fig. 5. Comparison of time and radial averaged solids velocity. F170 refers to Fluent and OF301 to OpenFOAM, A, B or C states the case, and x, y, or z state the direction of the velocity. The heights are A) 0.05, B) 0.35, C) 0.7, 1.00, D) 1.35, E) 1.75 m.

CONCLUSIONS

A laboratory-scale CFB unit was simulated with two CFD codes and the results of both codes were compared with measurements. While the aim was to utilized as similar settings as possible, there is some uncertainty about the model implementation and fully identical setups between the codes cannot be guaranteed. Simulation times required to complete the simulations were longer with Ansys Fluent than with OpenFOAM. Although Ansys Fluent allowed utilization of larger time steps than OpenFOAM, the OpenFOAM was faster.

OpenFOAM is not thoroughly documented but it is possible to see the source code to see the implementation, which is not as convenient as reading documentation and requires some knowledge in C++. In OpenFOAM, it is possible to modify solvers and implement new boundary conditions and models. On the other hand, only source of information for Ansys Fluent is the manual and verification of the used model equations is not feasible.

Both codes were able to reach reasonable good correspondence with the available pressure measurements. The mass flow to the cyclone was estimated well with Fluent, with larger differences for OpenFOAM compared to the measured values. In the averaged distributions of solids volume fraction and velocities of solids and gas, there were significant differences between the Fluent and OpenFOAM. From the results, it can be concluded that it is very likely that there are differences between the codes in models, their parameters or other solver parameters, which is causing the differences in the results, as they appear too large to be caused by the numerical methods. Without more detailed measurements, it is not possible to conclude which code offers more accurate results on the distributions profiles. Future attempts will be devoted to wider comparisons of simulations to the experiments and more detailed investigation of causes for the differences visible in the results.

REFERENCES

Ansys Inc. 2016a. Ansys Academic Research, Fluent 17.0. Released: 27th of January 2016.

Balogh, M., Parente, A., Benocci, C. 2012. RANS simulation of ABL flow over complex terrains applying an Enhanced k-e model and wall function formulation: Implementation and comparison for fluent and OpenFOAM. Journal of Wind Engineering and Industrial Aerodynamics 104-106, pp. 360-368.

Herzog, N., Schreiber, M., Egbers, C., Krautz, H. J. 2012. A comparative study of different CFD-codes for numerical simulation of gas–solid fluidized bed hydrodynamics. Computers and Chemical Engineering 39, pp. 41-46.

López, A., Nicholls, W., Stickland, M. T., Dempster, W. M. 2015. CFD study of Jet Impingement Test erosion using Ansys Fluent and OpenFOAM. Computer Physics Communications 197, pp. 88-95.

Niemi, T. 2012. Particle size distribution in CFD simulation of gas-particle flows, Master's Thesis. Aalto University.

OpenFOAM foundation Ltd. 2015. OpenFOAM 3.0.1. Released: 15th of December 2015, Available at: openfoam.org

Panday, R., Shadle, L. J., Shahnam, M., Cocco, R., Issangya, A., Spenik, J. S., Ludlow, J. C., Gopalan, B., Shaffer, F., Syamlal, M., Guenther, C., Reddy Karri, S. B., Knowlton, T. 2014. Challenge problem: 1. Model validation of circulating fluidized beds, Powder Technology 258, pp. 370–391.

Peltola, H. 2009. Dynamics in a circulating fluidized bed: experimental and numerical study. Master's thesis, Tampere University of Technology. 95 p.

Rusche, H. 2002. Computational fluid dynamics of dispersed two-phase flows at high phase fractions. PhD thesis, Imperial College, University of London. 343 p.

swak4Foam, 2016. Swiss Army Knife for OpenFOAM in the unofficial OpenFOAM wiki. Available at: http://openfoamwiki.net/index.php/Contrib/swak4Foam

Wang, Q., Niemi, T., Peltola, J., Kallio, S., Yang, H., Lu, F., Wei, L. Particle size distribution in CPFD modeling of gas–solid flowsin a CFB riser, Particuology 21, pp. 107–117.

Zhang, W., You, C. 2016. Numerical simulation of particulate flows in CFB riser with drag corrections based on particle distribution characterization, Chemical Engineering Journal 303, pp. 145–155.

Zhong, W., Xie, J., Shao, Y., Liu, X., Jin, B. 2015. Three-dimensional modeling of olive cake combustion in CFB, Applied Thermal Engineering 88, pp. 322-333.

Zeneli, M., Nikolopoulos, A., Nikolopoulos, N., Grammelis, P., Kakaras, E. 2015. Application of an advanced coupled EMMS-TFM model to a pilot scale CFB carbonator, Chemical Engineering Science 138, pp. 482–498.