9th International Freiberg Conference on IGCC & XtL Technologies
Closing the Carbon Cycle
3-8 June 2018
Berlin, Germany

CFD modeling of coal drying in impact dryer

Joanna Bigda[1]
[1]IChPW, Institute For Chemical Processing Of Coal Zabrze, Poland
email (Presenter): jbigda@ichpw.pl

In present work numerical studies for hydrodynamic of gas-solid flow and drying process have been carried out. There is little research on the simulation of the hydrodynamics and process of coal drying in impact dryers. Due to the fact that moisture content of hard coal is 20%, and brown coal of up to 70%, drying is necessary, in particular, before such processes, as briquetting, coking, gasification, combustion and others. With the increasing demand for better and higher quality products and for efficient operation, minimizing product degradation and energy consumption are current challenge for different applications.
Impact dryer is used for quick drying of bulk material (coal, lignite) in the turbulence chamber. The essence of drying in the dryer consists in the introduction of the drying material into the chamber perpendicularly to the stream of hot exhaust gases. The collision of two jets cause intense turbulence in the chamber. Turbulence and eddies increase contact of solid with hot gases and make better heat exchange and moisture evaporation. Therefore, the impact dryer system has the advantages of great drying intensity, short time, high efficiency, good quality and low energy consumption and is expected to have a wide application background and one of the most prospective drying for engineering applications.
This work presents the results of studies on determination of the influence of the impact dryer shape for coking coal drying on the effectiveness of the process in industrial scale (10 t/h). The present study has been undertaken in an effort to carry out a numerical study on the effect of the different position inlet (section) channel on the flow field and the dryer performance using CPFD Barracuda commercial solver. The applied software achieves very good results in modeling industrial scale installations, significantly reducing the time required for the modeling of highly concentrated systems (fluidized beds, dedusting in cyclones) comparing to other CFD programs.