Contribution ID: **143** Type: **Talk**

Modelling Dry Flue Gas Desulfurisation in a Circulating Fluidized Bed.

Monday, 3 December 2018 14:10 (20 minutes)

The burning of fossil fuels is identified as the biggest source of Sulfur dioxide (S02) emissions in South Africa. SO2 is a colourless gas with a nasty smell. It impacts both human health in the form of diminishing lung function and the environment through the effects of acid rain leading to deforestation. In order to meet minimum emissions standards for S02, as provided by the Air Quality Act of South Africa, big emitters such as Eskom and Sasol are required to implement Flue Gas Desulfurization (FGD) plants. Current FGD plants in South Africa are wet operated – that is, a sorbent like Calcium Hydroxide (Ca(OH)2) is mixed with water and sprayed as slurry into flue gas. In order to save water, dry FGD processes are considered. One such process is called Dry FGD using a Circulating Fluidized Bed (CFB). During this process dry Ca(OH)2 is introduced into a flue gas stream while water is separately introduced as a fine spray. SO2 is then reduced in a reaction involving the flue gas, Ca(OH)2 particles and water droplets.
The main reactor of a Dry FGD CFB plant is the vertical riser, which can typically be 3m in diameter and 20m high. The riser acts as a container in which the SO2 in the flue gas is exposed to the Ca(OH)2 particles and water droplets. Of importance are the residence time and the riser hold-up, which forms part of the overall successful design of a dry SO2 scrubber. Computational Fluid Dynamics is used to model the gas-solid two-phase mixture. The two-phase flow regime extends from dense to dilute turbulent flow, which renders the solution especially challenging. A Probability Density Function (PDF) approach is used to represent the microscale dynamics while filtering methods are used to yield practical solution on the scale of the plant.
Two codes are used for modelling the two-phase flows namely Neptune_CFD, which is the research code partly owned by EDF in France, and OpenFOAM, which allows the user to make fundamental changes in the source code. Both these codes are run at CHPC. The results of the study presented focus on transient runs performed on CHPC. Optimization of nodes and cores were done to yield the most effective solver parallelization for these codes. Hold-up and resident times of particles are predicted inside the riser. The mathematical models of the riser application of two-phase flows represent from the biggest computerized parallel models in this field.

Presenter Biography

Louis le Grange is a researcher in computational fluid dynamics at North-West University, Potchefstroom, South Africa. His interests include programming, fluid mechanics and in particular multiphase flows.

Primary author: Mr LE GRANGE, Louis (North-West University)

Presenter: Mr LE GRANGE, Louis (North-West University)

Session Classification: HPC Applications

Track Classification: Computational Mechanics