

CFD-Simulation of Siphone For primary aluminum production

Mohsen ameri siahooei¹, Darioush hossini roozbehani^{*2},

^{1,2}.Almahdi-Hormozal Aluminium Smelter, Bandar abass, P.O. Box:79171-7-6385, Iran

Abstract: Computational fluid dynamics (CFD) is a branch of fluid mechanics that uses numerical analysis and data structures to solve and analyze problems that involve fluid flows. Computers are used to perform the calculations required to simulate the interaction of liquids and gases with surfaces defined by boundary conditions. With high-speed supercomputers, better solutions can be achieved. Ongoing research yields software that improves the accuracy and speed of complex simulation scenarios such as transonic or turbulent flows. Initial experimental validation of such software is performed using a wind tunnel with the final validation coming in full-scale testing, e.g. flight tests. At the time of launching the Almahdi aluminum production line, the utility group faced many problems with metal depletion siphon for evacuating metal from pots, that these problems were mainly due to the lack of adequate suction of the siphon during the tapping.

In that time firstly, joints of the syphon door and the crucibles overflow were empirically sealed by changing the insulation type, and the size of the bolts and nuts of these joint was changed to use a power wrench for tightening the joint so that they can be sealed completely. However, one of the most important parts involved in the rate of the three way suction was the metal depletion siphon door, which was made of a three-way shape chamber, and two inlet and outlet nozzles. With spending large time and experimentally these parts were ultimately calibrated determined for the above mentioned purpose and their distance from each other in the three way was determined exactly. Then, the tapping operation was carried out in the normal way. Today mathematical tools are used in the further development and optimisation of crocyble. CFD (computational fluid dynamics)simulation is a tool which has been successfully established in manydiverse sectors of industry. It is the intention of this article to present the simulation of a siphone which has been done using the commercial software FLUENT. Numerical simulation shall provide the plant constructor with information on the guiding of flows, temperature distribution, heat input and heat losses in a siphone.

Keywords): “CFD-Simulation; primary aluminum production”