Simulation of a Sinter Cooler for Optimization and Design
Using CFD to Understand and Improve Cooling Process Efficiency

THE PROBLEM

The sinter cooler is a tall rotary cooler with permeable metallic hearth and it is used to cool the hot sinter particles before they can be charged into the blast furnace. The cooling efficiency of the sinter cooler is significant to productivity and the sinter quality. In addition, reducing cooling time by optimizing the operation conditions can save energy. The cooling efficiency of the sinter cooler depends upon the operating conditions and configuration of the cooler geometry. In this study, Computational Fluid Dynamics (CFD) had been employed to simulate the sinter cooler process. The parametric study was carried out to investigate the effects of operating conditions and evaluate the efficiency of an alternative design.

THE PROJECT

In this project, a CFD model has been developed and utilized to simulate the cooling process and examine the transient heat transfer characteristics of a sinter cooler. Temperature distributions within the sinter particles are considered in the model. The simulation results are comparable with industry observation. The parametric study includes cooling air flow rate, geometry, bed depth, initial sinter temperature, and size distribution of sinter particles. The simulation provides insight into the optimization of the cooling process and the improvement of product quality.

THE OUTCOME

The simulation results presented detailed flow patterns and transient heat transfer characteristics for different operating conditions and different geometry designs. The simulation results helped ArcelorMittal to solidify their decision to revamp the existing sinter cooler rather than build a new one and resulted in approximately $20 million in capital cost avoidance.