Abstract

Out of the many computational fluid dynamics (CFD) packages available, Fire Dynamics Simulator (FDS) has been the most popular CFD package for fire scenarios. However, the predictions of FDS or any CFD package depends on the implemented models within the package for different scenarios. Some of these models have been validated using real life experimental work while some have not. The aim of the work presented is to asses (validate) the model used for heat transfer between water droplets and solid surfaces using a set of experimental work performed using a superheated copper (Cu) plate. Calculation of the convective heat transfer coeffcient between the solid and the liquid phases was analysed for three different test cases.

The simulations were performed in a stepwise approach starting with simulations involving only the solid phase (Cu disc heated to 600 °C) and the water mist system individually before moving on to the simulations of the interaction of the water mist system and the Cu disc. Simulations were run using mono-disperse sprays due to the lack of information on the droplet size distributions.

The results show that FDS is indeed capable of producing relatively accurate results agreeing with the experimental work using mono-disperse sprays. This can be considered as the first step moving forward towards poly-disperse simulations where the droplet size distributions and the effects of the interaction of the plume and the water droplets can also be analysed.