CFD Modelling of Water Spray Impingement Cooling

Pedro Javier Vargas Rolón[1], Zoubir Acem[2], Rabah Mehaddi[2], Laurine Cappon[2], Tarek Beji[1*]

[1] Ghent University (UGent), Belgium
[2] LEMTA, Université de Lorraine, France
[*] Corresponding author, e-mail: Tarek.Beji@UGent.be

Bio

Tarek Beji obtained his Ph.D. degree from the University of Ulster (UK), in Fire Safety Engineering in 2009, with a thesis entitled 'Theoretical and experimental investigation on soot and radiation in fires'. In 2011, he joined Ghent University (UGent, Belgium) as a post-doctoral researcher and worked on the topic of fire forecasting. In 2014, he was awarded an FWO post-doctoral fellowship and worked for three years on 'numerical modelling of water sprays in fire-driven flows'. In October 2019, Tarek Beji was promoted Assistant Professor in Fire Dynamics at Ghent University (UGent) in the department of Structural Engineering and Building Materials where he is consolidating his research activities around the topic of 'numerical modelling of multi-phase aspects in fire-driven flows' with a particular interest in (*i*) water sprays, (*ii*) liquid pool fires, and (*iii*) soot. Tarek Beji is an experienced lecturer at UGent, giving courses in Fire Dynamics, Fluid Mechanics Applications in Fire, and Computational Fluid Dynamics (CFD) for Fire Safety Engineering.

Abstract

In fire dynamics, water spray surface cooling can be particularly efficient in (*i*) preventing the pyrolysis of combustible materials (to limit flame spread and fire growth) and (*ii*) avoiding excessive heating of structural elements, which can potentially lead to structural damage. The development of reliable numerical tools can therefore assist in the design of efficient mitigation measures. For that purpose, it is important to correctly model the convective heat transfer coefficient, h_w, of water droplets impinging onto the surface of a solid material.
It is in this context that numerical simulations of water spray surface cooling have been carried out with the Fire Dynamics Simulator (FDS 6.8.0), a Computational Fluid Dynamics (CFD) code developed by the National Institute of Standards and Technology (NIST, US). This is a

continuation of the validation work of Cédric Van de Vondel (UGent)[1], but relying on a new experimental campaign. The experimental configuration examined at the LEMTA laboratory at the University of Lorraine in France consists of a 1 m × 1 m plate, made of 3.1-mm thick steel, positioned horizontally and heated centrally from below by a radiant panel. During the heating phase, the steel surface temperature reached (at the center) steady-state values between 300 and 660°C, depending on the flow rate of the gas feeding the radiant panel. As opposed to a previous experimental campaign, the radiant panel is kept on and a 60° conical jet nozzle positioned at 0.5 m above the steel plate is activated, delivering a flow rate between 5 and 10 L/min, with a volume-median droplet diameter of about 220 to 245 µm.

It is found that the former default value of h_w = 300 W/(m^2·K) in FDS generally underestimates the cooling rate. An empirical correlation based on the local Nusselt number for a turbulent flow over a surface generally provides better results. Nevertheless, some challenges remain regarding specific test cases where, in the modelling, excessive water evaporation prior to impingement results in a significant cooling delay.

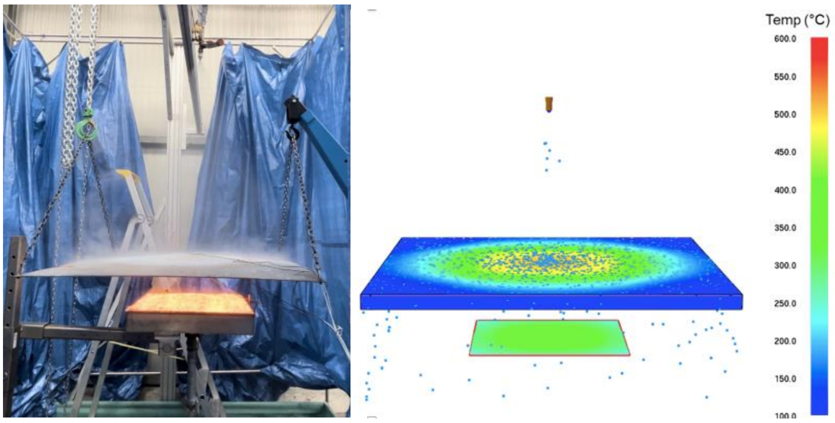

Figure 1 – Photo of the experimental set-up (left) and a snapshot of a CFD simulation (right).

[1] https://imfse.be/theses