Advanced Computational Fluid Dynamics Modelling of Water Sprays in Fire-Driven Flows

Martin Thielens

Summary

It is well-known that water-based fire suppression systems are an efficient means of mitigating the hazardous effects of a fire. However, there is no universal design of such systems. There is indeed a variety of systems (e.g., conventional sprinkler systems and water mist) and design parameters (e.g., water flow rate and nozzle positioning) which could depend on the specific application and the desired mechanisms to be put in place (e.g., smoke cooling, suppression and/or fuel surface wetting).

Although a lot of knowledge has been gained by experimental testing, there is still a need for a deeper understanding of the complex underlying physics behind the interaction of water sprays with fire-driven flows. This would allow the development of numerical tools with reliable predictive capabilities, supporting thus the design of water-based fire suppression systems in the framework of a performance-based approach or to illustrate equivalent safety level in case of deviations from the prescriptive rules.

The long-term objective of this work is to improve the current level of Computational Fluid Dynamics (CFD) capabilities for the application of interest by following a fundamental approach based on the analysis of 'simple' test cases. This approach allowed to focus on two specific sub-models.

The first sub-model is the heat-up of a single water droplet. The most common approach is to apply the infinite conductivity concept, i.e., uniform temperature distribution within the droplet. Whilst this approach is very well-suited (and justified) for small droplets, as typically encountered in a water mist, it might generate substantial uncertainties for large droplets. Therefore, a novel two-zone model has been implemented in an in-house code as well as in the Fire Dynamics Simulator (FDS). The analysis carried out shows that the distribution of the energy absorbed by the droplet, either by heat-up or by evaporation is affected by the choice of the heat transfer model. The latter also affects the amount of water vapor production and the amount of liquid water that reaches the floor. However, as mentioned the impact of the two-zone model is not really pronounced for small droplets that are typically found in water mist systems.

However, the second sub-model that has been investigated in this thesis is clearly connected to the application of interest of the IWMA as it is related to drag reduction in a dense spray. The uncertainty associated with the approach currently in FDS has been first identified in the PhD thesis of Yuanjun Liu [1]. In this former work, a comprehensive set of numerical simulations for the characterization of a water spray (called hereafter the Wuhan spray) in cold conditions has shown that the 'deficiency' in droplet drag reduction in the dense region close to the injection leads to a very narrow spray envelope in comparison to the experimental visualization and measurements. The approach adopted by Liu *et al.* in [2] for

purely diagnostic purposes is to set the drag coefficient to a constant value for all the droplets in the spray in order to reach a good agreement with the experimental data. In this PhD work, a more physical model, i.e., a Novel Drag Reduction (NDR) model has been implemented in FDS and assessed against the Wuhan spray data. It has shown to expand the spray envelope of the Wuhan spray by substantially reducing the drag force in dense regions (typically, near the nozzle) of the spray. The simulation results were closer to the experimental observations when the NDR model is implemented than when the default FDS code is used. However, the results with the NDR model have shown to be sensitive to the mesh size. The newly proposed drag reduction model, has been assessed against more sprays (in cold conditions) in order to examine its ability to cope with different levels of density. The NDR model does not really affect or deteriorate the simulation results for more dilute sprays. Besides, the configuration of spray penetration in a hot air jet, which has been examined experimentally in [3] and extensively simulated at Ghent University, has been revisited in this PhD. It is argued that the difficulties to correctly predict the position of the stagnation plane in the previous numerical studies can be attributed to some (if not large) extent to a deficiency in drag reduction modelling (which has not been considered before). The current results show a better prediction of the interface height between the air jet and the water mist. The preliminary results are very promising but should be deepened in a future work.

Finally, a clear strategy has been set up during this PhD thesis in order to ensure a good dissemination as well as an easy access to the models and the codes developed during this work. This is an essential element for the continuity of the work that is clearly in line with the long-term objective of the thesis which is to improve the predictive capabilities of CFD modelling in the simulation of water spray interaction with fire-driven flows. Two sharing and collaborating platforms have been used, namely GitHub and Open Science Framework (OSF). The first one has been used to clone and fork the FDS source code developed by the National Institute of Standards and Technology (NIST). This has offered the possibility to directly implement and test the novel sub-models in a more advanced CFD tool. In addition to that, the OSF platform has been used in order to host the in-house codes and the modified versions of the FDS code such that it could be further used by other researchers. This is an evolutive platform that could be used to continue the work initiated here and enhance it.

To summarize, the main novelties and added values of this work are : (1) the fundamental approach, based on the analysis of 'simple' test cases, that has allowed examining the limitations of several sub-models that are commonly used and developing novel models to potentially overcome those limitations; (2) in-house codes that have been developed to assess correlations in simplified codes where the complexity has been reduced to its minimum and to offer the possibility to focus on the physical aspect of interest; (3) the implementation, at a second stage, of the novel models in the source code of FDS which has allowed assessing those models in a more advanced CFD tool and (4) the definition and the implementation of a clear Research Data Management (RDM) plan that warrants the findability and the accessibility of the codes in order to ensure the continuation of the work as performed during the thesis.

Bibliography

[1] Y. Liu, The Combined Effect of Water Mist System and Longitudinal Ventilation on Fire and Smoke Behavior in Tunnel Fires, Ghent University & Wuhan University: Doctoral dissertation, 2021, https://biblio.ugent.be/publication/8710084.

[2] Y. Liu, T. Beji, M. Thielens, Z. Tang, Z. Fang and B. Merci, "Numerical Analysis of a Water Mist Spray: the Importance of Various Numerical and Physical Parameters, Including the Drag Force," Fire Safety Journal, vol. 127, p. 103515, 2022, https://doi.org/10.1016/j.firesaf.2021.103515.

[3] X. Zhou, "Characterization of Interactions Between Hot Air Plumes and Water Sprays for Sprinkler Protection," Proc. Combust. Inst., vol. 35, pp. 2723-2729, 2015, https://doi.org/10.1016/j.proci.2014.05.078.