Investigation of high-pressure water mist impact on tunnel fire: full-scale experimental validation of numerical modelling

Jakub Bielawski[1,3], Bogdan Racięga[2], Xinyan Huang[3], Wojciech Węgrzyński[1]

[1]Building Research Institute, Poland, j.bielawski@itb.pl, w.wegrzynski@itb.pl

[2]Baltic Fire Laboratory, Poland, bogdan.raciega@bafilab.com

[3]Hong Kong Polytechnic University, China, xy.huang@polyu.edu.hk

BIO

Jakub Bielawski is a research assistant at the Building Research Institute in Poland and a visiting PhD student at Hong Kong Polytechnic University. His PhD research focuses on performance-based adaptation of fire safety systems in terms of actual conditions airflow, fire behavior, and smoke flow. In his research, he combines experimental and numerical methods with novel analysis methods such as machine learning algorithms. Jakub is part of the team of the Building Research Institute for the commissioning of fire safety systems of actual tunnels in Poland. He performed more than 150 hot smoke tests in tunnels and buildings. He also participates in research projects in the field of fire safety of building facades, green walls, and timber structures and aerodynamic tests in wind tunnel.

Background

Vehicle fires in transportation infrastructure are characterized by high severity for structures and hazards for humans. The length of road tunnels could be up to several kilometers, which means that a large number of passengers at the same time could be exposed to fire hazards. A relevant component for the improvement of the fire safety strategy in the tunnel is the application of a fixed firefighting system. These include high-pressure water mist (HPWM) systems, which have been successfully used for fire protection in tunnels. Due to the benefits associated with hydraulic installation, it can be used for both new tunnels and existing tunnels undergoing retrofitting. Current requirements for confirming the effectiveness of a fire suppression system, such as HPWM, are related to full-scale fire tests. As a consequence, there are limitations to fire scenarios performed and the spatial geometry of the test tunnel.

Objectives

The main objective of the study is to develop a numerical model of the performance of HPWM suppression of tunnel fires and compare it with the results of full-scale test fires. Validation of the CFD model will enable monitoring changes in the flow parameters of smoke and hot fire gases as well as the temperature of the tunnel structures. Achieving good convergence between experimental and numerical results can be the basis for extrapolating the model for other tunnel geometries and boundary conditions.

Methodology

Full-scale experiments were carried out in the Applus+ TST test tunnel in San Pedro, Spain. The facility is dedicated to performing large fires with fire suppression systems. Ultrafog HPWM system were tested in reference to the research project SOLIT[2]. This investigation covered cases of 60 MW test fires at moderate and low ventilation velocities, 3 m/s and 1.5 m/s, respectively.

The numerical study was performed using Computational Fluid Dynamics (CFD) code ANSYS Fluent which is based on the Finite Volume Method for resolving governing equations of continuity, momentum, energy, and other sub-models of physical phenomenon for fluid flow and heat transfer problems. The internal geometry of the tunnel was built and discretized into a numerical mesh with polyhedral elements. The fire source was incorporated as a diesel pool fire with the Eddy-Dissipation Model for the combustion process. The Heat Release Rate (HRR) value was assumed from data obtained experimentally. Water mist was modelled using the Discrete Phase Model (DPM) based on Lagrangian particles formulation. Spacing of the HPWM nozzles and their characteristics of droplet size distribution and spray pattern were introduced according to experimental setup and results.

Results and conclusions

The results of experiments and numerical simulations indicate the effectiveness of using HPWM for tunnel fire suppression. In both cases, critical environmental values in terms of temperature and radiation heat flux at the target points were not exceeded. Additionally, a positive effect on the reduction of back-layering was observed in both methods. The final presentation outcome will consist of a quantitative results for temperature profiles, spatiotemporal pressure and velocity evolution as well as heat absorbed by water mist.

KEYWORDS: numerical modelling, CFD-DPM, high-pressure water mist, tunnel fire, fire suppression